FAGOR AUTOMATION

CNC 800 M

New Features (Ref.0204in)

Version 2.1 (July 1995)

1. P627(1). DIVIDING FACTOR FOR ELECTRONIC HANDWHEEL FEEDBACK SIGNALS

Machine parameter P627(1) is used with P612(6), P626(6) and P627(6) which indicate the multiplying factor for the electronic handwheel feedback signals for the X, Y and Z axis respectively.

Machine parameter P627(1) indicates whether all handwheel feedback signals are to be divided or not.
P627(1)=0 They are not divided.
P627(1)=1 All handwheel feedback signals are divided by two.

Examples for the X axis so the CNC assumes 100 pulses/turn with 25, 50 and 100 line handwheels:

25 line Fagor handwheel:	P612(6)=0 and P627(1)=0	25 x 4 / 1 = 100 lines
50 line Fagor handwheel:	P612(6)=1 and P627(1)=0	50 x 2 / 1 = 100 lines
100 line Fagor handwheel:	P612(6)=1 and P627(1)=1	100 x 2 / 2 = 100 lines

Version 2.4 (June 1996)

1. HANDWHEELS AFFECTED BY FEED-HOLD.

Until now, it was assumed that electronic handwheels operated like mechanical handwheels and, therefore, were not affected by Feed-hold.

However, some applications require the electronic handwheels (manual pulse generators) to be affected by Feed-hold.

Machine parameter "P628(2)" indicates whether they are or not affected by Feed-hold.

P628(2) = 0 Not affected
P628(2) = 1 Affected.

2. AUTOMATIC TOOL CHANGERS (ATC)

With this feature, it is possible to manage tool changers at any time.

Until now, this was only possible while executing a program (P99996) in Automatic Mode.

Parameter setting:

Machine parameter "P628(3)" indicates whether the machine has an ATC or not.

P628(3) = 0 No ATC.
P628(3) = 1 The machine has an ATC.

In either case, the CNC considers machine parameter "P743" and "P745" .

P743 Standard subroutine to be executed prior to a T function
P745 Standard subroutine to be executed after a T function

The subroutines associated with the T function must contain the tool selection sequence and must be defined by the manufacturer in one of the special ISO-coded user programs: P99994 or P99996.

Both subroutines are defined by an integer between 0 and 89.
If set to 0, the CNC assumes that no subroutine is to be executed.

2.1 MACHINE WITH MANUAL TOOL CHANGER

Machine parameter "P628(3)" must be set to "0" (no ATC available).

Basic operation in JOG or DRO mode

Every time a new tool is selected, (T?? - START), the CNC acts as follows:

1.- If machine parameter "P743" is set to a value other than "0", the CNC executes that standard subroutine.

2.- The CNC outputs the BCD code and assumes the new tool.

3.- If machine parameter "P745" is set to a value other than "0", the CNC executes that standard subroutine.

Basic operation while executing an automatic operation

Every time the execution of an automatic operation requires a tool change, (T01 active and the cycle requests T02), the CNC behaves as follows:

1.- If machine parameter "P743" is set to a value other than "0", the CNC executes that standard subroutine.

2.- It displays the message: "TOOL CHANGE" and interrupts program execution.

3.- When the operator presses the **[CYCLE START]** key, the CNC outputs the BCD code and assumes the new tool.

4.- If machine parameter "P745" is set to a value other than "0", the CNC executes that standard subroutine.

5.- The CNC resumes the execution of the automatic operation.

Basic operation while executing the ISO-coded user program (99996)

a) One or both machine parameters "P743" and "P745" have been set to a value other than "0".

Every time the execution of the ISO program (99996) requires a tool changer, the CNC behaves as follows:

1.- If machine parameter "P743" is set to a value other than "0", the CNC executes that standard subroutine.

2.- The CNC outputs the BCD code and assumes the new tool.

3.- If machine parameter "P745" is set to a value other than "0", the CNC executes that standard subroutine.

4.- The CNC resumes program execution.

b) Both machine parameters "P743" and "P745" have been set to "0".

Every time the execution of the ISO program (99996) requires a tool changer, the CNC behaves as follows:

1.- The CNC outputs the BCD code and assumes the new tool.

2.- It executes the internal standard subroutine N99, which:

Displays the message: "TOOL CHANGE"
and interrupts program execution (M00).

3.- When the operator presses the **[CYCLE START]** key, the CNC resumes program execution.

2.2 MACHINE WITH AUTOMATIC TOOL CHANGER (ATC)

Machine parameter "P628(3)" must be set to "1" (ATC available).

Basic operation in JOG or DRO mode

Every time a new tool is selected, (T?? - START), the CNC acts as follows:

1.- If machine parameter "P743" is set to a value other than "0", the CNC executes that standard subroutine.

2.- The CNC outputs the BCD code and assumes the new tool.

3.- If machine parameter "P745" is set to a value other than "0", the CNC executes that standard subroutine.

Basic operation while executing an automatic operation

Every time the execution of an automatic operation requires a tool change, (T01 active and the cycle requests T02), the CNC behaves as follows:

1.- If machine parameter "P743" is set to a value other than "0", the CNC executes that standard subroutine.

2.- The CNC outputs the BCD code and assumes the new tool.

3.- If machine parameter "P745" is set to a value other than "0", the CNC executes that standard subroutine.

4.- The CNC resumes the execution of the automatic operation.

Basic operation while executing the ISO-coded user program (99996)

a) One or both machine parameters "P743" and "P745" have been set to a value other than "0".

Every time the execution of the ISO program (99996) requires a tool changer, the CNC behaves as follows:

1.- If machine parameter "P743" is set to a value other than "0", the CNC executes that standard subroutine.

2.- The CNC outputs the BCD code and assumes the new tool.

3.- If machine parameter "P745" is set to a value other than "0", the CNC executes that standard subroutine.

4.- The CNC resumes program execution.

b) Both machine parameters "P743" and "P745" have been set to "0".

Every time the execution of the ISO program (99996) requires a tool changer, the CNC behaves as follows:

1.- The CNC outputs the BCD code and assumes the new tool.

2.- It executes the internal standard subroutine N99, which:

Displays the message: "TOOL CHANGE"
and interrupts program execution (M00).

3.- When the operator presses the **[CYCLE START]** key, the CNC resumes program execution.

3. TREATMENT OF THE M19 (SPINDLE ORIENT)

When using ATC, the spindle must be oriented before changing tools.

This features implements function M19 to manage spindle orientation.

M19 should be included in the standard subroutine to be executed before the T function (machine parameter P743).

Requirements:

The spindle must have a spindle encoder installed.

This encoder must be connected via connector "A5", which is the same one used for the electronic handwheel associated with the Z axis.

To use this feature on machines having a handwheel associated with the Z axis, connector "A5" must be shared by the handwheel and the spindle encoder.

Precautions on machines having a Z axis handwheel:

- Both feedback devices must be commutated (handwheel and spindle encoder).
- The CNC interprets the feedback signals at connector "A5" as follows:

  In "Spindle Orient" mode" (M19) as spindle feedback.
  In "Open Loop Spindle" mode (M3, M4, M5) as handwheel pulses.
- If the spindle switches from "Spindle Orient" mode to "Open Loop" mode without swapping the feedback device at connector "A5", the CNC will take the spindle pulses as handwheel pulses.

Parameter setting:

Machine parameter "P800" indicates whether there is a spindle encoder installed or not and, consequently, whether "Spindle Orientation" is available or not.

P800	= 0	No spindle encoder installed. "Spindle Orient" not available.
P800	<>0	Spindle encoder line count (number of pulses/rev).

Besides having an spindle encoder (P800 other than 0), the following machine parameters must also be set:

P609(2)	Spindle counting direction
P700	Spindle speed when operating in M19
P601(7)	Sign of the spindle analog output associated with M19.
P612(8)	Type of spindle encoder reference mark (home).
P619(6)	Spindle orient in both directions (Negative S also possible).
P719	Minimum analog spindle output when in M19.
P717	Spindle in-position zone when in M19
P718	Proportional gain K of the spindle when in M19
P916	Spindle Orient position when executing M19 without an "S" value.

Programming format

Spindle Orient is programmed as: 'M19 S4.3', where:

M19	Indicates that the spindle is now moving in Closed loop.
S4.3	Indicates the number of degrees it has to move from the reference zero mark

Programming format while in DRO mode

To orient the spindle, proceed as follows:

* Press the keystroke sequence: **[F] - [BEGIN] - [END]**
* The bottom of the CNC screen shows the letter "**M**"
* Key in **[1] - [9] - [S] - (desired value) - [CYCLE START]**

Basic operation

A "M19 S4.3" type block is executed as follows:

* The CNC outputs the M19 code as any other "M" function so the electrical cabinet can execute it.

* If the spindle was in open loop (M3, M4), the CNC slows the spindle down until its speed is below the value set by machine parameter "P700" and, then, homes the spindle.

* The CNC orients the spindle to the preset position (S4.3) and at the speed set by machine parameter "P700".

  If a block containing only an "M19" is executed (without "S4.3"), the CNC orients the spindle to the position set by machine parameter "P916". If "P916=0", the spindle spins indefinitely at the rpm set for M19.

  The orienting direction is set by machine parameter "P601(7)"; however, the spindle may be oriented in either direction by using machine parameter "P619(6)".

* The spindle will remain in closed loop until:

  - An M3, M4 or M5 is executed
  - An S ???? is executed
  - A Reset is carried out
  - An M30 is executed
  - An execution error comes up

Example:

M3 S1000	Spindle in open loop and turning clock-wise.
M19	Spindle in closed loop, home search and orientation to position indicated by parameter "P916".
M19 S100	Orient to 100° (from reference mark, home)
S1000	Spindle in open loop recovering its previous turning direction (M3):
M19 S200	Spindle in closed loop, home search and orientation to 200º from reference mark.

1. GENERATING AN ISO-CODED PROGRAM

With this CNC, the ISO code (low level) for an operation or a part-program may be generated.

To use this feature, machine parameter "P630(1)" must be set to "1".

This ISO program always has the number: 99996 and can be stored either at the CNC or at a PC.

Program 99996 is a special user program in ISO code and can be:
Generated from an operation or a part-program.
Edited at the CNC itself via menu option: "Auxiliary Modes - Edit program 99996"
Loaded into the CNC after being generated at a PC.

Generating the ISO program (99996) at the CNC.

This CNC has 11 K of memory space to store program 99996. If the generated program is larger than that, the CNC will issue the relevant error message.

To generate program 99996, proceed as follows:

* If it is an operation, select or define the desired operation.
* If it is a part-program, select the desired one in the part-program directory and place the cursor on its header ("PART 01435". A listing of the operations it consists of must appear).
* Press the keystroke sequence: [CALC] [7]. The CNC will show the graphic simulation screen.
* Press . The CNC starts simulating the part and generating its ISO-coded program 99996.
* When done with the simulation, program 99996 stored in CNC memory will contain all simulated blocks in ISO code.

Generating the ISO program (99996) at a PC

Usually, the 99996 program generated from a part-program exceeds the available memory space of the CNC.

By using "DNC30", this program may be generated at a PC.

To do this, proceed as follows:

* Activate DNC communications and execute the DNC30 program at the PC.
* Select at the PC the menu option: "Program Management - Receive Digitizing".
* At the CNC, select the operation or place the cursor on the part-program header ("PART 01435"). A listing of the operations it consists of must appear).
* Press [CALC] [8]. The CNC will display the graphic simulation screen.
* Press . The CNC starts simulating the part and generating program 99996.
* When done with the simulation, the 99996 program generated at the PC will contain all the blocks simulated by the CNC in ISO code.

  This program can be executed at the CNC through the menu option: "Execute infinite program" of the DNC30.

Note: While generating the ISO-coded program, no tool compensation is applied when simulating. However, the generated program will have the corresponding G41 and G42.

2. RIGID TAPPING IS NOW AVAILABLE

From this version on, typical tapping is possible (with a clutch) "P630(3)=0" as well as rigid tapping "P630(3)=1".

For rigid tapping, the CNC must control the spindle checking its turning speed at all times and supplying the necessary analog voltage for the spindle to turn at the selected speed.

General considerations:

Rigid tapping is an interpolation between the spindle and the Z axis.
The acceleration and deceleration time for the spindle and the Z axis should be the same.
The following error (lag) of the spindle and that of the Z axis must be proportional. For example, if when tapping at F1000mm/min, S1000rpm (pitch=1mm) the observed following error is Z=1mm and S=360º, it means that they are both perfectly synchronized.
A machine parameter is now available to adjust the spindle's response (acc/dec) for each spindle range (gear).
Since the Z axis gain is different for machining and for rigid tapping, the CNC offers 2 parameters, one for each case.
Output THREADING_ON (I97) is active when carrying out a rigid tapping.

Machine parameters related to the spindle:

P800	Number of spindle encoder pulses (0...9999)
P601(7)	Sign of the analog S associated with M19 (0 or 1)
P609(2)	Spindle counting direction (0 or 1)
P612(8)	Type of spindle home marker pulse (0 = -, 1 = +)
P719	Minimum spindle analog voltage (0...255) P719=0 ==> 2,5mV P719=10 ==> 25.0mV (10 x 2.5) P719=1 ==> 2,5mV P719=255 ==> 637.5mV (255 x 2.5)
P717	In-position zone (dead-band) of the spindle. Number of encoder pulses (0...255) The CNC internally applies a x4 multiplying factor to the pulses coming from the encoder. Thus, with a 1000 line encoder and P717= 100, the in-position zone will be: (360°/4000)x100= ±9°
P718	Proportional spindle gain K (0...255) It sets the analog voltage corresponding to a following error of 1 spindle encoder pulse. Analog (mV.) = P718 x Following error (pulses) x 2.5 mV / 64
P751, P747, P748, P749	Duration of the spindle acc/dec ramp in ranges 1, 2, 3, 4 (0...255) Value 1=20 ms
P746	Feed-forward gain in rigid tapping (0...255)
P750	Z axis proportional gain K1 in rigid tapping (0...255)
P625(1)	The tap entry is synchronized with the spindle home marker pulse (0=No, 1=Yes)

Feedback inputs:

P630(4)=0 Connector A5 is used for spindle feedback and for the Z axis handwheel.
Both feedback devices must be switched externally.

P630(4)=1 Connector A5 is only used for spindle feedback.
Connector A6 is used for the X axis handwheel.
Connector A4 is used for Y and Z axes handwheels.
PLC output O46 indicates which axis moves when turning the handwheel connected to A4.
"O46=0" for the Y axis and "O46=1" for the Z axis.

ISO Programming

It is programmed by means of function G33 (threading) indicating axis feedrate and spindle speed.

Examples: G33 Z -10 F1000 S1000 M3
F1000 S1000 M3
G33 Z-10

Functions G00, G01, G02 and G03 cancel function G33.

3. CURRENT CNC SOFTWARE VERSION

From this version on, when accessing the EPROM checksum screen [Auxiliary Modes] [Special Modes] [8]

The CNC will show the checksum of each EPROM and the current CNC software version. For example: Version 3.1

4. SCREEN SAVER

When machine parameter “P626(7)=1” the screen saver function acts as follows:

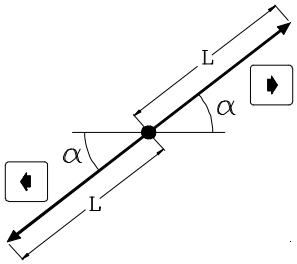
After 5 minutes without pressing any key or the CNC not receiving any new data for refreshing (updating) the screen, it goes blank. Video is restored when pressing any key or when the CNC receives data to update the screen.

5. SEMIAUTOMATIC LINEAR MILLING

To access this mode, select the linear milling mode and press [LEVEL +] to get into semiautomatic mode.

This operation cannot be stored as in a part-program.

The path angle (α) and length (L) must be defined.



Jog the axes with the handwheel up to the starting point and press the corresponding JOG key (just press it once, it does not have to be held down).

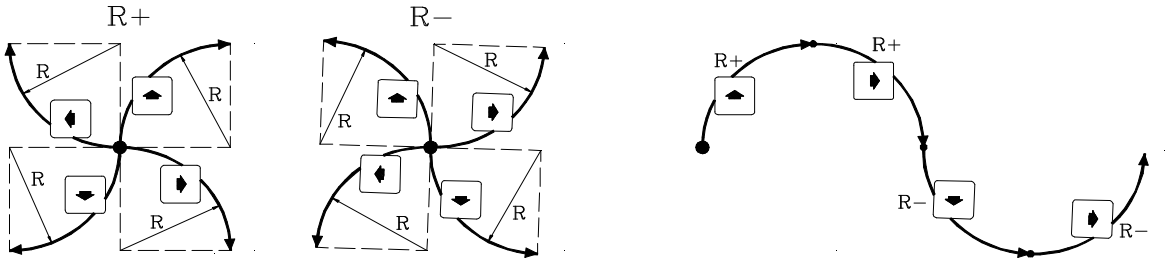
The axes will travel the indicated distance "L" in the indicated direction and at an angle "α" or until the [ ] key is pressed.

6. SEMIAUTOMATIC ARC MILLING

To access this mode, select the arc milling mode and press [LEVEL +] to get into semiautomatic mode.

This operation cannot be stored in a part-program.

The rounding radius (R) must be defined. Its sign indicates the turning direction (R+ and R-)



Jog the machines with the handwheels to the desired starting point and press the corresponding JOG key (just press it once, it does not have to be held down). The machine will make a 90º arc in the indicated direction.

7. CROSS COMPENSATION

Besides compensating for measuring errors due to inaccurate leadscrews (leadscrew error), this CNC offers cross compensation in order to compensate for errors caused by one axis onto another. A typical case would be beam (ram) sag compensation.

To use cross compensation, one must define the axis to be compensated and the one inflicting the error onto the other one when moving.

Machine parameters related to cross compensation:

P623(1)	Cross compensation applied to the X axis (0=No, 1=Yes)
P620(5)	Cross compensation applied to the Y axis (0=No, 1=Yes)
P620(4)	Cross compensation applied to the Z axis (0=No, 1=Yes)
P623(2), P623(3)	Axis inflicting the error onto the other one.

Affected (compensated) axis			
	P623(1)	P620(5)	P620(4)
X	1	0	0
Y	0	1	0
Z	0	0	1

Moving ("guilty") axis		
	P623(3)	P623(2)
X	0	1
Y	1	0
Z	1	1

Examples:

Compensate Y for Z axis movement	P620 (* * * 1 0 * * *)	P623 (* * * * * 0 0 0)
Compensate X for Y axis movement	P620 (* * * 0 0 * * *)	P623 (* * * * * 1 0 1)

8. FUNCTION M80 WHEN "Z" AS A DRO AXIS

This feature is available when the Z axis is set to work as a DRO axis. "P617(4)=1".

Whenever the Z axis has to be moved, the CNC shows the text: "Act upon Z".
Also, from this version on, it executes the auxiliary function M80. With this function, it is possible to act upon the hydraulic or mechanical device that controls the Z axis.

9. MACHINE SAFETY REGULATION

This CNC offers the following features to comply with machine safety regulations.

Enabling of the CYCLE START key *from the PLC*

This feature is available when machine parameter "P630(5)=1"

PLC output O25 indicates whether the CYCLE START key is enabled (=1) or not (=0)

Axes movements controlled by Feed-Hold. (It was already available)

Feed-Hold input, pin 15 of connector I/O 1, must be normally high.

If while moving the axes, the Feed-Hold input is brought low, the CNC keeps the spindle turning and stops the axes with 0V or velocity command (analog signal) and keeping their enables ON.

When this signal is brought back up, the CNC will resume the movement of the axes.

Axes jogging feedrate limited by PLC.

This feature is available when machine parameter "P630(5)=1"

When activating PLC output O26, the CNC assumes the feedrate set by machine parameter "P814"

Handwheel managed by the PLC.

Machine parameter "P628(2)" indicates whether the axes movements with handwheels are affected by Feed-Hold (=1) or not (=0)

Machine parameter "P630(2)" indicates whether the multiplying factor indicated by the MFO switch position is applied (=0) or the one indicated by the PLC outputs O44 and O45 (=1) (already available)

O44	O45	
0	0	According to switch setting
1	0	Same as x1 setting of the switch
0	1	Same as x10 setting of the switch
1	1	Same as x100 setting of the switch

Spindle control from the PLC.

This feature is available when "P630(5)=1"

Output O27 =1 "tells" the CNC to apply the spindle analog voltage set by the PLC. The value of this analog signal is set at register R156 and sent to the CNC by mark M1956.

R156= 0000 1111 1111 1111 => +	10V.	R156= 0001 1111 1111 1111 => -	10V.
R156= 0000 0111 1111 1111 => +	5V.	R156= 0001 0111 1111 1111 => -	5V.
R156= 0000 0011 1111 1111 => +	2,5V.	R156= 0001 0011 1111 1111 => -	2,5V.
R156= 0000 0000 0000 0000 => +	0V.	R156= 0001 0000 0000 0000 => -	0V.

Also, PLC output O43, lets you control the rotation of the spindle. (Already available).
It must be normally low.
If it is brought up, the CNC stops the spindle.
When it is brought back up, the CNC restarts the spindle.

Information for the PLC on the status of the machine reference (home) search

I88 Home search in progress.
I100 X axis home search done.
I101 Y axis home search done.
I102 Z axis home search done.

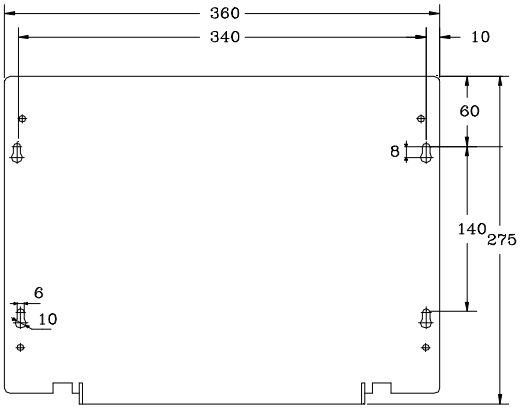
R120 The lower half of this register indicates the last key pressed.
This value is maintained for 200 milliseconds unless another key is pressed before then.
This register can be canceled from the PLC after being processed.

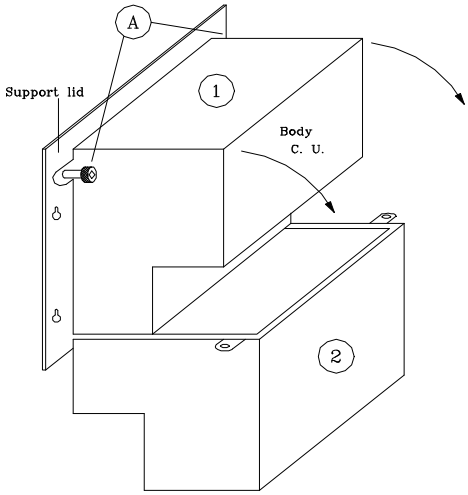
R121
- bit 1 Indicates that the Milling operation is selected (=1)
- bit 2 Indicates that the Positioning operation is selected (=1)
- bit 3 Indicates that the Pocket Milling operation is selected (=1)
- bit 4 Indicates that the Boss Milling operation is selected (=1)
- bit 5 Indicates that the Corner Roughing operation is selected (=1)
- bit 6 Indicates that the Surface Milling operation is selected (=1)
- bit 7 Indicates that one of the machining operations (Center punching, Drilling, etc.) is selected (=1)
- bit 8 Indicates that the "Auxiliary Modes" option is selected (=1)
- bit 9 Indicates that the "Tool Calibration" option is selected (=1)
- bit 10 Indicates that the "Graphic Simulation" mode is selected (=1)
- bit 16 Indicates that the mode relevant to following cycle parameters: "finising pass, finishing feedrate, finishing tool and safety distances on X and Z" is selected (=1)

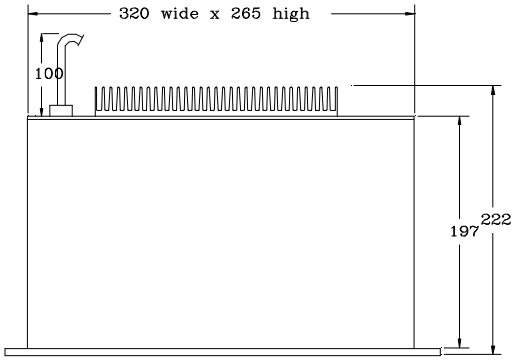
1. MODULAR CNC

The modular 800M CNC consists of the Central Unit module (CPU), the Monitor and the keyboard.

Central Unit. It is usually located in the electrical cabinet and is mounted by means the holes it has for this purpose on its support lid. Dimensions in mm.





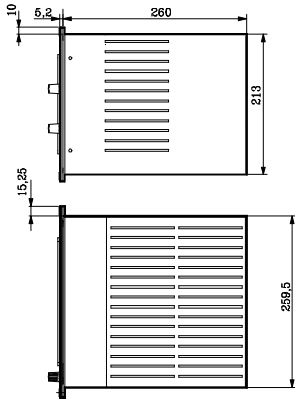


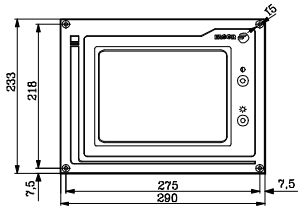
When installing it, observe enough clearance to swing the Central Unit open for future access to its interior.

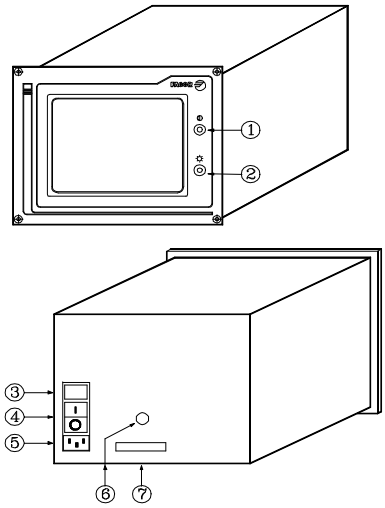
To swing the central unit open, undo the two knurled nuts at the top and swing it open while holding its body.

Monitor. It may be mounted anywhere on the machine, preferably at operator's eye level.

9" Amber and 10" Color Monitor

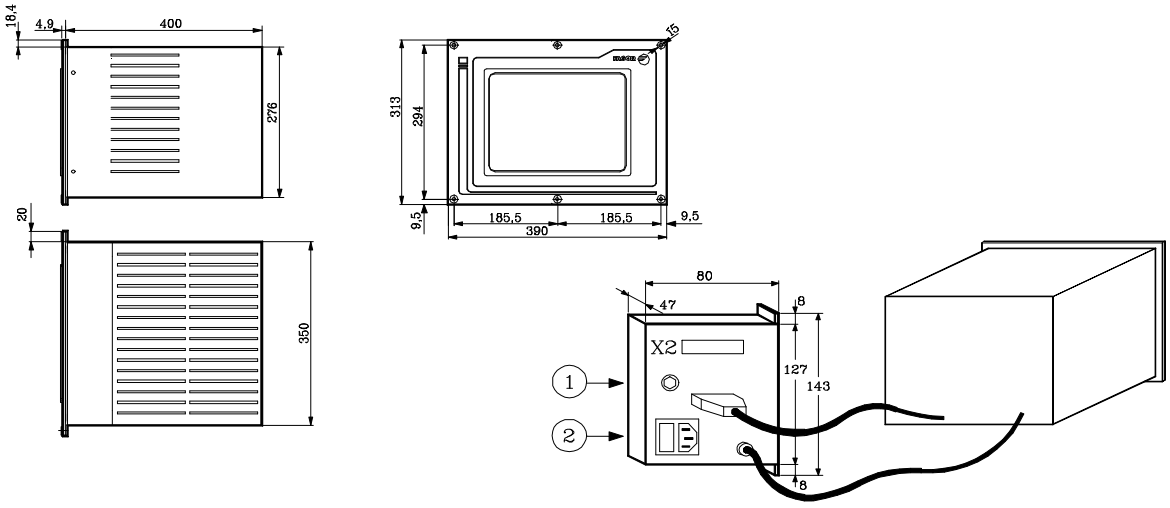






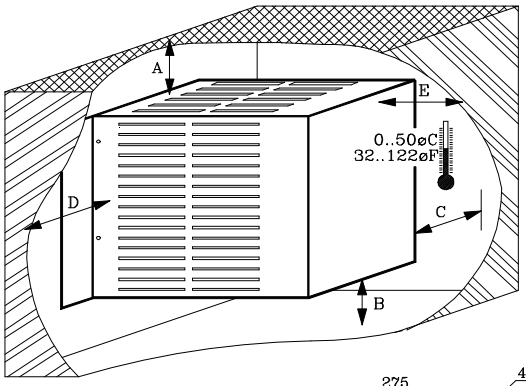
1.- Contrast
2.- Brightness
3.- Two 3.15A/250V fast fuses (F), one per mains line, to protect the mains input.
4.- Power switch.
5.- 220 Vac mains and ground connection.
6.- General ground connection terminal. Metric 6mm
7.- 15-pin SUB-D type male connector for connecting it with the Central Unit.

14" Color Monitor

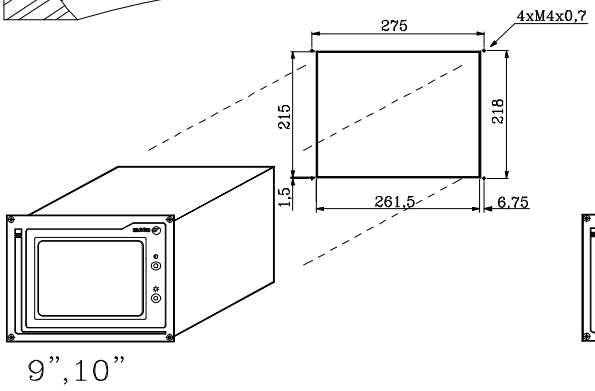


X2 15-pin SUB-D type male connector for connecting it with the Central Unit.
1.- General ground connection terminal. Metric 6mm
2.- 220 Vac mains and ground connection.

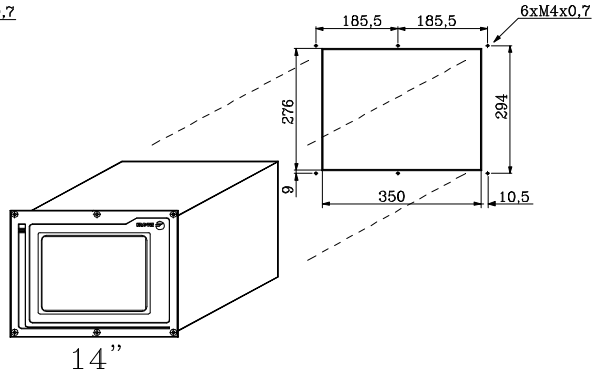
Monitor enclosures.



	A, B, C, D	E
9" & 10 Monitor	25 mm	150 mm
14" Monitor	100 mm	50 mm

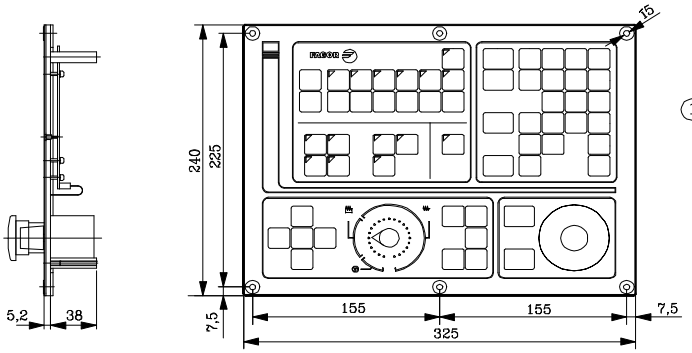


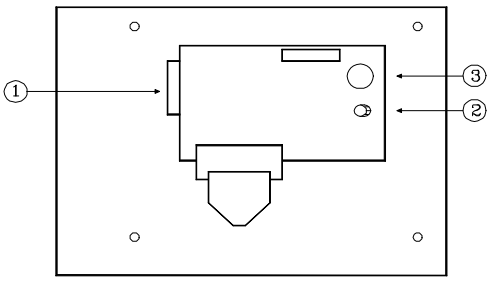
9",10"



14"

Keyboard It may be mounted anywhere on the machine.





Rear panel

1.- 25-pin SUB-D type female connector for connecting it with the Central Unit.
2.- Buzzer volume adjusting potentiometer.
3.- Buzzer.

Connector for connecting the Central Unit with the Monitor.

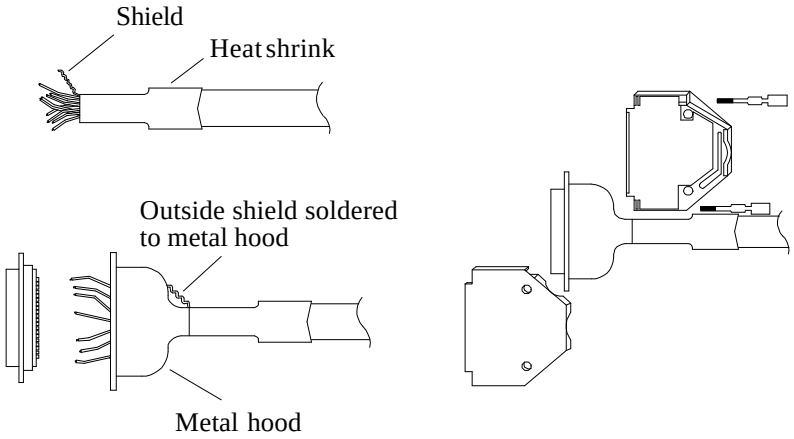
FAGOR AUTOMATION provides the cable required for this connection. It comes with a 15-pin SUB-D type male connector at one end and a 15-pin SUB-D type female connector at the other.

Both connectors have a latching system UNC4.40 by means of two screws.

The supplied cable has 6 twisted pairs of 0.34 mm² wires (6 x 2 x 0.34mm²), with overall shield and acrylic cover. It has a specific impedance of 120 Ohms. Its maximum length must be 25 meters (82 feet).

The cable shield is soldered to the metal hoods (housings) of both connectors and connected to pin 1 at both the Central Unit and Monitor/keyboard connectors.

PIN	SIGNAL
1	GND
2	H
3	V
4	I
5	R
6	G
7	B
8	Not connected
9	Not connected
10	H
11	V
12	I
13	R
14	G
15	B
Metal hood	Shield



Connector for connecting the Central Unit with the keyboard.

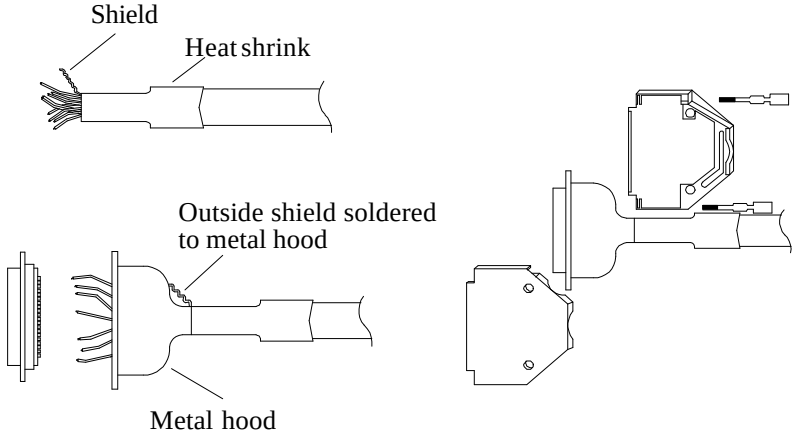
FAGOR AUTOMATION provides the cable required for this connection. It comes with a 25-pin SUB-D type male connector at each end.

Both connectors have a latching system UNC4.40 by means of two screws.

The supplied cable has 25 wires of 0.14 mm² (25 x 0.14mm²), with overall shield and acrylic cover. Its maximum length must be 25 meters (82 feet).

The cable shield is soldered to the metal hoods (housings) of both connectors and connected to pin 1 at both the Central Unit and Monitor/keyboard connectors.

PIN	SIGNAL
1	GND
2	C9
3	C11
4	C13
5	C15
6	C1
7	C3
8	C5
9	C7
10	D1
11	D3
12	D5
13	D7
14	C8
15	C10
16	C12
17	C14
18	C0
19	C2
20	C4
21	C6
22	D0
23	D2
24	D4
25	D6
Metal hood	Shield



2. PROGRAMMING IN ISO CODE. NEW FUNCTION F34

P1 = F34 Parameter P1 takes the value of the tool causing the call to the subroutine associated with the tools. Do not mistake it with function F24 which returns the number of the tool currently being used.

3. PROGRAMMING IN ISO CODE. RIGID TAPPING

When carrying out rigid tapping in 800M mode, the CNC acts as follows:

1.- Internally generates function M81 (switching feedback)
2.- Carries out rigid tapping.
3.- Internally generates function M82 (switching back to previous feedback)

Therefore, when programming rigid tapping in ISO code, function M81 must be programmed in block preceding rigid tapping and function M82 in the one following it.

4. 1000 LINE ENCODER AS A 1250 LINE ENCODER

With this feature, the CNC can use a 1000 line encoder as it were a 1250 line encoder.

P630(6) X axis 1000 line encoder as 1250 line encoder (0=No, 1=Yes)
P630(7) Y axis 1000 line encoder as 1250 line encoder (0=No, 1=Yes)
P630(8) Z axis 1000 line encoder as 1250 line encoder (0=No, 1=Yes)

A typical case: When using a motor with 1000 line encoder on a 5 mm pitch ballscrew.

The necessary calculations for setting axis resolution will be made using the selected line count (1000 or 1250 pulses).

5. PLCI. INPUT I104

When the Feedrate Override Switch on the operator panel is set on one of the handwheel positions (x1, x10, x100), input I104 is set to "1".

6. PLCI. R120 AND THE KEY

From this version on, even when the is disabled by parameter P618(1), PLCI register R120 contains its code when it is pressed.

Version 3.04 (March 2002)

1. TOOL COMPENSATION CANCELATION

Sometimes, it may be interesting to move the tool to a set position without applying its length compensation.
In these cases, program "T.0" and the CNC will acts as follows:

- It does not change the tool (it does not call its associated subroutine).
- It cancels its associated offset (assuming a zero length and radius compensation).

The "T.xx" instruction may be programmed at any time, even inside the program P99996 or in its associated subroutine. The CNC assumes the new "xx" offset. When programming "T.0", it assumes a zero tool length and radius compensation.

2. FEEDBACK SIGNAL DIVIDING FACTOR

Parameters P631(8), P631(7), P631(6), P631(5) and P631(4) are used with parameters P604(8), P604(7), P604(6), P604(5) and P616(8) that indicate the multiplying factor for the feedback signals of the X, Y, Z, W, V axes respectively.

X axis	Y axis	Z axis	W axis	V axis
P604(8)	P604(7)	P604(6)	P604(5)	P616(8)
P631(8)	P631(7)	P631(6)	P631(5)	P631(4)

They indicate whether the feedback signals are divided (=1) or not (=0).

P631(8)=0, P631(7)=0, P631(6)=0, P631(5)=0 and P631(4)=0 They are not divided
P631(8)=1, P631(7)=1, P631(6)=1, P631(5)=1 and P631(4)=1 They are divided by 2.

Example: To obtain a 0.01 mm resolution using a square-wave encoder mounted on the X axis whose pitch is 5 mm/turn.

Nr of pulses = Leadscrew pitch / (Multiplying factor x Resolution)

If P604(8)=0 & P631(8)=0	x4 multiplying factor	Nr of pulses = 125
If P604(8)=1 & P631(8)=0	x2 multiplying factor	Nr of pulses = 250
If P604(8)=0 & P631(8)=1	x2 multiplying factor	Nr of pulses = 250
If P604(8)=1 & P631(8)=1	x1 multiplying factor	Nr of pulses = 500

3. FEEDBACK FACTOR.

The resolution of the axis is determined by the leadscrew pitch and the number of pulses of the encoder mounted on the motor. Sometimes, the resolution resulting from the available leadscrew / encoder combination does not match any of the resolution values allowed for the machine parameters (1, 2, 5, 10 microns or ten-thousandths of an inch).

Example: With a 6 mm/turn leadscrew pitch and a 2500 line encoder, the resulting resolution values are:
Resolution = Leadscrew pitch / (Nr of encoder pulses x multiplying factor).
With x1 multiplying factor 2.4 micron resolution
With x2 multiplying factor 1.2 micron resolution
With a x4 multiplying factor 0.6 micron resolution

A new axis machine parameter is now available to solve these cases and it is referred to as Feedback Factor in order to adapt the resulting resolution to the existing setup.
P819 Feedback factor for the X axis P820 Feedback factor for the Y axis P821 Feedback factor for the Z axis
Values between 0 and 65534, a "0" value means that this feature is not being used.

Use the following formula to calculate the "Feedback factor":
Feedback Factor = (Gear ratio x Leadscrew Pitch / Encoder pulses) x 8.192

Examples:						
	Gear ratio	1	1	2	1	
	Leadscrew Pitch	4000	6000	6000	8000	(microns)
	Encoder	2500	2500	2500	2500	(pulses/turn)
	Feedback factor	13,107.2	19,660.8	39,321.6	26,214.4	

The machine parameters only admit integers, but the Feedback Factor sometimes may have decimals. In those cases, set the machine parameter to the integer part of that value and use the leadscrew error compensation table to make up for the decimal part.

The values for this table are calculated using the following formula:
Leadscrew position = Leadscrew error (microns) x Integer portion of feedback factor /Decimal portion of feedback factor

In this case: Gear ratio = 1 Pitch = 6000 Encoder = 2500
Feedback factor = 19,660.8 Machine parameter = 19660
For leadscrew error of 20 microns Leadscrew position = 20 x 19,660 / 0.8 = 491,520
The following table is obtained by using this calculation.

Leadscrew position		Amount of error at that position	
P0 =	-1966,000	P1 =	-0.080
P2 =	-1474,500	P3 =	-0.060
P4 =	-983,000	P5 =	-0.040
P6 =	-491,500	P7 =	-0.020
P8 =	0	P9 =	0
P10 =	491,500	P11 =	0.020
P12 =	983,000	P13 =	0.040
P14 =	1472,500	P15 =	0.060
P16 =	1966,000	P17 =	0.080

Headquarters (SPAIN): **Fagor Automation S. Coop.**
Bº San Andrés s/n, Apdo. 144
E-20500 Arrasate - Mondragón
Tel: +34-943-719200/039800
Fax: +34- 943-791712
+34-943-771118 (Service Dept.)
www.fagorautomation.com
E-mail: info@fagorautomation.es

FAGOR CNC 800M

INSTALLATION MANUAL

Ref. 9705 (in)

ABOUT THE INFORMATION IN THIS MANUAL

This manual is addressed to the machine manufacturer.

It includes the necessary information for new users as well as advanced subjects for those who are already familiar with the 800M CNC product.

It may not be necessary to read this whole manual. Consult the list of "New Features and Modifications" and the appendix related to the machine parameters. Practically all of them are cross-referenced indicating the chapter and section of the manual where they are described.

This manual explains all the functions of the 800M CNC family. Consult the Comparison Table for the models in order to find the specific ones offered by your CNC.

To install the CNC onto your machine, we suggest that you consult the appendix regarding the enclosures required to mount the CNC as well as chapter 1 (CNC configuration) which indicates the CNC dimensions and details the pin-out of its connectors.

Chapter 2 (Power and machine interface) shows how to connect the CNC to power A.C. (Mains) and to the electrical cabinet.

Chapter 3 "Auxiliary Functions" shows how to access special operating modes.

To adapt the CNC to the machine, set the CNC machine parameters. We suggest that you consult chapters 4, 5, 6 and the appendices related to the machine parameters listed in numerical order.

Both appendices offer cross references indicating the section of the manual describing each parameter.

When explaining each parameter in detail, chapters 4, 5 and 6, they sometimes refer to chapter 7 (concepts) where some of them are dealt with in further detail indicating how to perform various adjustments of the CNC-machine interface.

Once all machine parameters are set, we suggest that you write their settings down on the charts provided for this purpose in the appendix on "Machine Parameter Setting Chart".

There is also an appendix on error codes which indicates some of the probable reasons which could cause each one of them.

Notes: The information described in this manual may be subject to variations due to technical modifications.

FAGOR AUTOMATION, S.Coop. reserves the right to modify the contents of the manual without prior notice.

INDEX

Section Page

Comparison Table for Fagor 800T CNC models ix
New Features and modifications xi

INTRODUCTION

Declaration of Conformity 3
Safety Conditions 4
Warranty Terms 7
Material Returning Terms 8
Additional Remarks 9
Fagor Documentation for the 800M CNC 10
Manual Contents 11

Chapter 1 CONFIGURATION OF THE CNC

1.1 Introduction 1
1.2 Dimensions and installation 2
1.3 Connectors and interface 3
1.3.1 Connectors A1, A2, A3, A4 5
1.3.1.1 Dip-switches for connectors A1, A2, A3, A4 6
1.3.2 Connector A5 7
1.3.2.1 Dip-switches for connector A5 8
1.3.3 Connector A6 9
1.3.4 RS232C connector 10
1.3.5 Connector I/O 1 13
1.3.5.1 Inputs of connector I/O 1 14
1.3.5.2 Outputs of connector I/O 1 17
1.3.6 Connector I/O 2 19
1.3.6.1 Outputs of connector I/O 2 20

Chapter 2 POWER AND MACHINE INTERFACE

2.1 Power interface 1
2.1.1 Internal power supply 1
2.2 Machine interface 2
2.2.1 General considerations 2
2.2.2 Digital outputs 4
2.2.3 Digital inputs 4
2.2.4 Analog outputs 5
2.2.5 Feedback inputs 5
2.3 Set-up 6
2.3.1 General considerations 6
2.3.2 Precautions 6
2.3.3 Connection 7
2.3.4 System input/output test 8
2.4 Emergency input/output connection 10
2.5 Activation / Deactivation of external devices 13

Section Page

Chapter 3 AUXILIARY FUNCTIONS

3.1 Millimeters / Inches 1
3.2 Tool length compensation 1
3.3 Tool Table 2
3.3.1 Modification of tool dimensions 3
3.4 Tool calibration 4
3.5 Execution / Simulation of program P99996 5
3.5.1 Execution of program P99996 5
3.5.1.1 Tool inspection 6
3.5.1.2 Execution modes 7
3.5.1.3 CNC reset 7
3.5.1.4 Displaying program blocks 7
3.5.1.5 Display modes 8
3.5.2 Simulation of program 99996 10
3.5.2.1 Zoom function 11
3.6 Auxiliary modes 12
3.7 Special modes 12
3.7.1 Test 13
3.7.2 General parameters 15
3.7.3 Decoded "M" functions 16
3.7.3.1 M functions sent out in BCD 18
3.7.4 Leadscrew error compensation 19
3.8 Peripherals 21
3.8.1 Peripheral mode 21
3.8.2 DNC communications 22
3.9 Lock / unlock 23
3.10 Editing program P99996 24

Chapter 4 MACHINE PARAMETERS

4.1 Introduction 1
4.2 Operating with parameter tables 2
4.3 General machine parameters 3
4.3.1 Input/output parameters 5
4.3.2 Parameters related to the handwheels 10
4.3.3 Parameters related to the operating mode 12
4.3.4 Parameters for the RS232C serial line 16

Chapter 5 MACHINE PARAMETERS FOR THE AXES

5.1 Parameters related to axis resolution 2
5.2 Parameters related to the analog outputs 5
5.3 Parameters related to travel limits 6
5.4 Feedrate related parameters 7
5.5 Parameters related to axis control 9
5.6 Parameters related to machine reference zero 11
5.7 Parameters for acceleration/deceleration of the axes 13
5.7.1 Linear acceleration/deceleration 13
5.7.2 Bell-shaped acceleration/deceleration 14
5.7.3 Feed-forward gain 15
5.8 Leadscrew related parameters 16
5.8.1 Leadscrew backlash 16
5.8.2 Leadscrew error 17
5.9 Special machine parameters 19

Section Page

Chapter 6 SPINDLE MACHINE PARAMETERS

6.1 Parameters related to spindle speed range change 1
6.2 Parameters for analog spindle speed output 2
6.3 Parameters for spindle speed output in BCD 3

Chapter 7 CONCEPTS

7.1 Feedback systems 1
7.1.1 Counting frequency limits 2
7.2 Movement by electronic handwheel 3
7.3 Axis resolution 4
7.4 Adjustment of the axes 10
7.4.1 Adjustment of the drift (offset) and maximum feedrate (G00) 11
7.4.2 Gain adjustment 13
7.4.3 Proportional gain adjustment 14
7.4.3.1 Calculation of K1, K2 and gain break-point 16
7.4.4 Feed-Forward gain adjustment 18
7.4.4.1 Calculation of feed-forward gain 18
7.4.5 Leadscrew error compensation 19
7.5 Reference systems 21
7.5.1 Reference points 21
7.5.2 Machine reference (home) search 22
7.5.3 Adjustment of the value corresponding to the machine reference point (home) 23
7.5.4 Software travel limits for the axes 24
7.5.5 Considerations about the machine reference point 25
7.6 Spindle 26
7.6.1 Spindle speed range change 29
7.7 Feedhold, transfer inhibit and M-done signal processing 31
7.8 Auxiliary functions M, S, T 32
7.8.1 Decoded M function table 33
7.8.2 M, S, T function transfer 34
7.8.3 M, S, T function transfer using the M-done signal 35

APPENDICES

A Technical characteristics of the CNC 2
B Enclosures 4
C CNC inputs and outputs 5
D 2-digit BCD coded "S" output conversion table 6
E Machine parameter summary chart 7
F Sequential machine parameter list 10
G Machine parameter setting chart 15
H Decoded "M" function setting chart 17
I Leadscrew error compensation setting chart 18
J Maintenance 19

ERROR CODES

COMPARISON TABLE FOR FAGOR 800M CNC MODELS

AVAILABLE 800M CNC MODELS

	800-MG	800-MGI
X, Y axes control	l	l
Z axis as DRO	l	l
Controlled Z axis	l	l
Spindle	l	l
Tools	99	99
Tool Radius Compensation	l	l
Tool Length Compensation	l	l
Electronic Handwheels	3	3
RS 232C Communications	l	l
Integrated PLC (PLCI)		l
ISO-coded program editing (P99996)	l	l
Execution of ISO-coded program (P99996)	l	l
Graphics	l	l

NEW FEATURES AND MODIFICATIONS

Date: **July 1995** ***Software version:*** **2.1 and newer**

FEATURE	AFFECTED MANUAL AND SECTION	
Clear all arithmetic parameter contents setting them to "0".	Installation Manual Operating Manual	Section 3.9 Section 3.8&6.9
ISO Programming.	Programming Manual	
Editing of program P99996 at the CNC.	Installation Manual Operating Manual	Section 3.10 Section 3.9
When interrupting execution, the keys for the spindle, the coolant and for O1, O2, O3 and TOOL are enabled.	Installation Manual Operating Manual Operating Manual	Section 3.5.1 Section 2.5.1 Section 6.5
Subroutine associated to the execution of a tool (only when executing program P99996)	Installation Manual Programming Manual	Section 4.3 Chapter 9.
ISO codes of the 800T CNC	Programming Manual	

Date: **November 1995** ***Software version:*** **2.2 and newer**

FEATURE	AFFECTED MANUAL AND SECTION	
Subroutines to be executed before and after the "T" function.	Installation Manual Programming Manual	Section 4.3 Chapter 9
"M" functions associated with automatic operations.	Operating Manual	Section 4.1.2
"M" functions associated with machining operations.	Operating Manual	Section 5.1.1

INTRODUCTION

Atention:

Before starting up the CNC, carefully read the instructions of Chapter 2 in the Installation Manual.

The CNC must not be powered-on until verifying that the machine complies with the "89/392/CEE" Directive.

DECLARATION OF CONFORMITY

Manufacturer: Fagor Automation, S. Coop.

Barrio de San Andrés s/n, C.P. 20500, Mondragón -Guipúzcoa- (ESPAÑA)

We hereby declare, under our responsibility that the product:

Fagor 800M CNC

meets the following directives:

SAFETY:

EN 60204-1 Machine safety. Electrical equipment of the machines.

ELECTROMAGNETIC COMPATIBILITY:

EN 50081-2 Emission
- EN 55011 Radiated. Class A, Group 1.
- EN 55011 Conducted. Class A, Group 1.
- EN 61000-3-2 Current Harmonics
- EN 61000-3-3 Voltage fluctuations and flickers

EN 50082-2 Immunity
- EN 61000-4-2 Electrostatic Discharges.
- EN 61000-4-3 Radiofrequency Radiated Electromagnetic Fields.
- EN 61000-4-4 Bursts and fast transients.
- EN 61000-4-5 Conducted high voltage pulses in mains (Surges)
- EN 61000-4-6 Conducted disturbance induced by radio frequency fields.
- EN 61000-4-8 Magnetic fields at mains frequency
- EN 61000-4-11 Voltage fluctuations and Outages.
- ENV 50204 Fields generated by digital radio-telephones

As instructed by the European Community Directives: on Low Voltage 73/23/CEE, on Machine Safety 89/392/EEC, 89/336/EEC on Electromagnetic Compatibility and its upgrades.

In Mondragón, on October 1st, 2001

Fagor Automation, S. Coop. Ltda.
Director Gerente

Fdo.: Julen Busturia

SAFETY CONDITIONS

Read the following safety measures in order to prevent damage to personnel, to this product and to those products connected to it.

This unit must only be repaired by personnel authorized by Fagor Automation.

Fagor Automation shall not be held responsible for any physical or material damage derived from the violation of these basic safety regulations.

Precautions against personal damage

Use proper Mains AC power cables

To avoid risks, use only the Mains AC cables recommended for this unit.

Avoid electrical overloads

In order to avoid electrical discharges and fire hazards, do not apply electrical voltage outside the range selected on the rear panel of the Central Unit.

Ground connection

In order to avoid electrical discharges, connect the ground terminals of all the modules to the main ground terminal. Before connecting the inputs and outputs of this unit, make sure that all the grounding connections are properly made.

Before powering the unit up, make sure that it is connected to ground

In order to avoid electrical discharges, make sure that all the grounding connections are properly made.

Do not work in humid environments

In order to avoid electrical discharges, always work under 90% of relative humidity (non-condensing) and 45º C (113º F).

Do not work in explosive environments

In order to avoid risks, damage, do not work in explosive environments.

Precautions against product damage

Working environment

This unit is ready to be used in Industrial Environments complying with the directives and regulations effective in the European Community

Fagor Automation shall not be held responsible for any damage suffered or caused when installed in other environments (residential or homes).

Install the unit in the right place

It is recommended, whenever possible, to instal the CNC away from coolants, chemical product, blows, etc. that could damage it.

This unit complies with the European directives on electromagnetic compatibility. Nevertheless, it is recommended to keep it away from sources of electromagnetic disturbance such as.

- Powerful loads connected to the same AC power line as this equipment.
- Nearby portable transmitters (Radio-telephones, Ham radio transmitters).
- Nearby radio / TC transmitters.
- Nearby arc welding machines
- Nearby High Voltage power lines
- Etc.

Enclosures

The manufacturer is responsible of assuring that the enclosure involving the equipment meets all the currently effective directives of the European Community.

Avoid disturbances coming from the machine tool

The machine-tool must have all the interference generating elements (relay coils, contactors, motors, etc.) uncoupled.

Use the proper power supply

Use an external regulated 24 Vdc power supply for the inputs and outputs.

Grounding of the power supply

The zero volt point of the external power supply must be connected to the main ground point of the machine.

Analog inputs and outputs connection

It is recommended to connect them using shielded cables and connecting their shields (mesh) to the corresponding pin (See chapter 2).

Ambient conditions

The working temperature must be between +5° C and +45° C (41ºF and 113º F)
The storage temperature must be between -25° C and 70° C. (-13º F and 158º F)

Main AC Power Switch

This switch must be easy to access and at a distance between 0.7 m (27.5 inches) and 1.7 m (5.6 ft) off the floor.

Protections of the unit itself

It carries two fast fuses of 3.15 Amp./ 250V. to protect the mains AC input.

All the digital inputs and outputs have galvanic isolation via optocouplers between the CNC circuitry and the outside.

They are protected by an external fast fuse (F) of 3.15 Amp./ 250V. against reverse connection of the power supply.

Precautions during repair

Do not manipulate the inside of the unit

Only personnel authorized by Fagor Automation may manipulate the inside of this unit.

Do not manipulate the connectors with the unit connected to AC power.

Before manipulating the connectors (inputs/outputs, feedback, etc.) make sure that the unit is not connected to AC power.

Safety symbols

Symbols which may appear on the manual

WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

Symbols that may be carried on the product

WARNING. symbol
It has an associated text indicating those actions or operations may hurt people or damage products.

"Electrical Shock" symbol
It indicates that point may be under electrical voltage

"Ground Protection" symbol
It indicates that point must be connected to the main ground point of the machine as protection for people and units.

WARRANTY TERMS

WARRANTY

All products manufactured or marketed by Fagor Automation has a warranty period of 12 months from the day they are shipped out of our warehouses.

The mentioned warranty covers repair material and labor costs, at FAGOR facilities, incurred in the repair of the products.

Within the warranty period, Fagor will repair or replace the products verified as being defective.

FAGOR is committed to repairing or replacing its products from the time when the first such product was launched up to 8 years after such product has disappeared from the product catalog.

It is entirely up to FAGOR to determine whether a repair is to be considered under warranty.

EXCLUDING CLAUSES

The repair will take place at our facilities. Therefore, all shipping expenses as well as travelling expenses incurred by technical personnel are NOT under warranty even when the unit is under warranty.

This warranty will be applied so long as the equipment has been installed according to the instructions, it has not been mistreated or damaged by accident or negligence and has been manipulated by personnel authorized by FAGOR.

If once the service call or repair has been completed, the cause of the failure is not to be blamed the FAGOR product, the customer must cover all generated expenses according to current fees.

No other implicit or explicit warranty is covered and FAGOR AUTOMATION shall not be held responsible, under any circumstances, of the damage which could be originated.

SERVICE CONTRACTS

Service and Maintenance Contracts are available for the customer within the warranty period as well as outside of it.

MATERIAL RETURNING TERMS

When returning the CNC, pack it in its original package and with its original packaging material. If not available, pack it as follows:

1.- Get a cardboard box whose three inside dimensions are at least 15 cm (6 inches) larger than those of the unit. The cardboard being used to make the box must have a resistance of 170 Kg (375 lb.).

2.- When sending it to a Fagor Automation office for repair, attach a label indicating the owner of the unit, person to contact, type of unit, serial number, symptom and a brief description of the problem.

3.- Wrap the unit in a polyethylene roll or similar material to protect it.

When sending the monitor, especially protect the CRT glass.

4.- Pad the unit inside the cardboard box with poly-utherane foam on all sides.

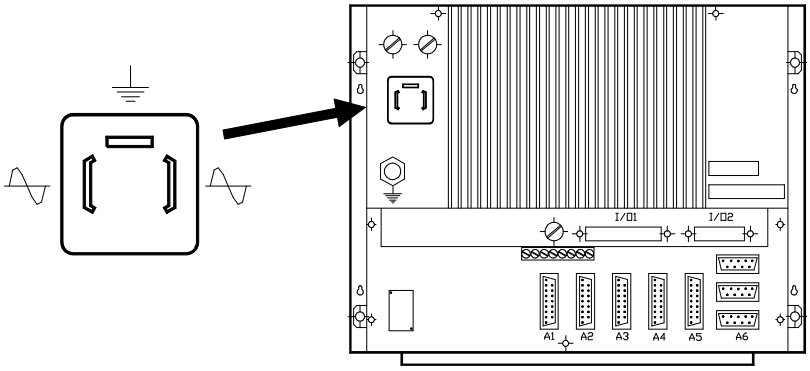
5.- Seal the cardboard box with packing tape or industrial staples.

ADDITIONAL REMARKS

* Mount the CNC away from coolants, chemical products, blows, etc. which could damage it.

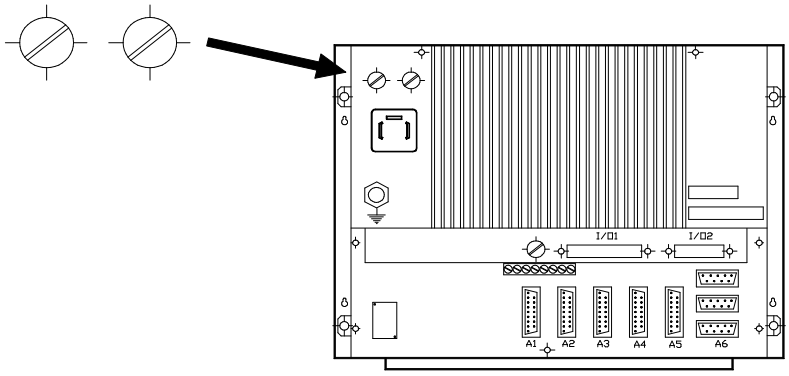
* Before turning the unit on, verify that the ground connections have been properly made. See Section 2.2 of this manual.

* To prevent electrical shock use the proper mains AC connector. Use 3-wire power cables (one for ground connection)



* Before turning the unit on, verify that the external AC line fuse, of each unit, is the right one.

  Must be 2 fast fuses (F) of 3.15 Amp./ 250V.



* In case of a malfunction or failure, disconnect it and call the technical service. Do not manipulate inside the unit.

FAGOR DOCUMENTATION FOR THE 800M CNC

800M CNC OEM Manual Is directed to the machine builder or person in charge of installing and starting up the CNC.

It has the Installation manual inside. Sometimes, it may contain an additional manual describing New Software Features recently implemented.

800M CNC USER Manual Is directed to the end user or CNC operator.

It contains 2 manuals:
Operating Manual describing how to operate the CNC.
Programming Manual describing how to program the CNC.

Sometimes, it may contain an additional manual describing New Software Features recently implemented.

DNC 25/30 Software Manual Is directed to people using the optional DNC communications software.

DNC 25/30 Protocol Manual Is directed to people wishing to design their own DNC communications software to communicate with the 800 without using the DNC25/30 software..

PLCI Manual To be used when the CNC has an integrated PLC.

Is directed to the machine builder or person in charge of installing and starting up the PLCI.

DNC-PLC Manual Is directed to people using the optional communications software: DNC-PLC.

AUTOCAD 8050 Manual Is directed to people wishing to design their own customized CNC screens and symbols on AUTOCAD. This manual indicates how to set up the Autocad program for the CNC to correctly interpret the designed screens and symbols.

FLOPPY DISK Manual Is directed to people using the Fagor Floppy Disk Unit and it shows how to use it.

MANUAL CONTENTS

The installation manual consists of the following sections:

Index

Comparative Table for Fagor 800M CNC models

New Features and modifications

Introduction
- Warning sheet prior to start-up
- Declaration of Conformity
- Safety Conditions
- Warranty terms
- Shipping conditions
- Additional remarks
- Fagor documents for the 800M CNC
- Manual Contents

Chapter 1 CNC configuration
- Indicates the CNC dimensions
- Detailed description of all the connectors.

Chapter 2 Power and machine connection.
- Indicates how to connect it to Main AC power.
- Ground connection.
- Characteristics of the digital inputs and outputs.
- Characteristics of the analog output.
- Characteristics of the feedback inputs
- CNC setup and start-up
- System I/O testing
- Connection of the Emergency input and output.
- How to activate and deactivate external devices.

Chapter 3 Auxiliary functions.
- Indicates how to select the work units (mm/inches).
- How to define the tool table.
- How to calibrate and inspect a tool.
- How to run a system test.
- How to access the machine parameters.
- How to access and operate with the decoded "M" functions.
- How to apply leadscrew error compensation.
- How to operate with peripherals.
- How to lock and unlock the machine parameters and the program memory.
- How to edit, execute and simulate program 99996.

Chapter 4 Machine parameters.
- How to operate with machine parameters.
- How to set the machine parameters.
- Detailed description of the general machine parameters.

Chapter 5 Machine parameters for the axes.
- Detailed description of the machine parameters for the axes.

Chapter 6 Machine parameters for the spindle.
- Detailed description of the machine parameters for the spindle.

Chapter 7 Concepts.
- Feedback systems, resolution
- Adjustment of the axes and their gains.
- Reference Systems: Reference systems, search and setting
- Software travel limits for the axes.
- Acceleration / deceleration.
- Spindle: speed control and range change.
- "Feed Hold" and "M-done" signal processing (treatment)
- Auxiliary M, S, T function transfer

Appendix
- A CNC technical characteristics.
- B Enclosures.
- C CNC inputs and outputs.
- D 2-digit BCD coded spindle "S" output
- E Machine parameter summary chart
- F Sequential machine parameter listing
- G Machine parameter setting chart
- H Decoded "M" function chart
- I Leadscrew error compensation table
- J Maintenance

Error Code

1. CONFIGURATION OF THE CNC

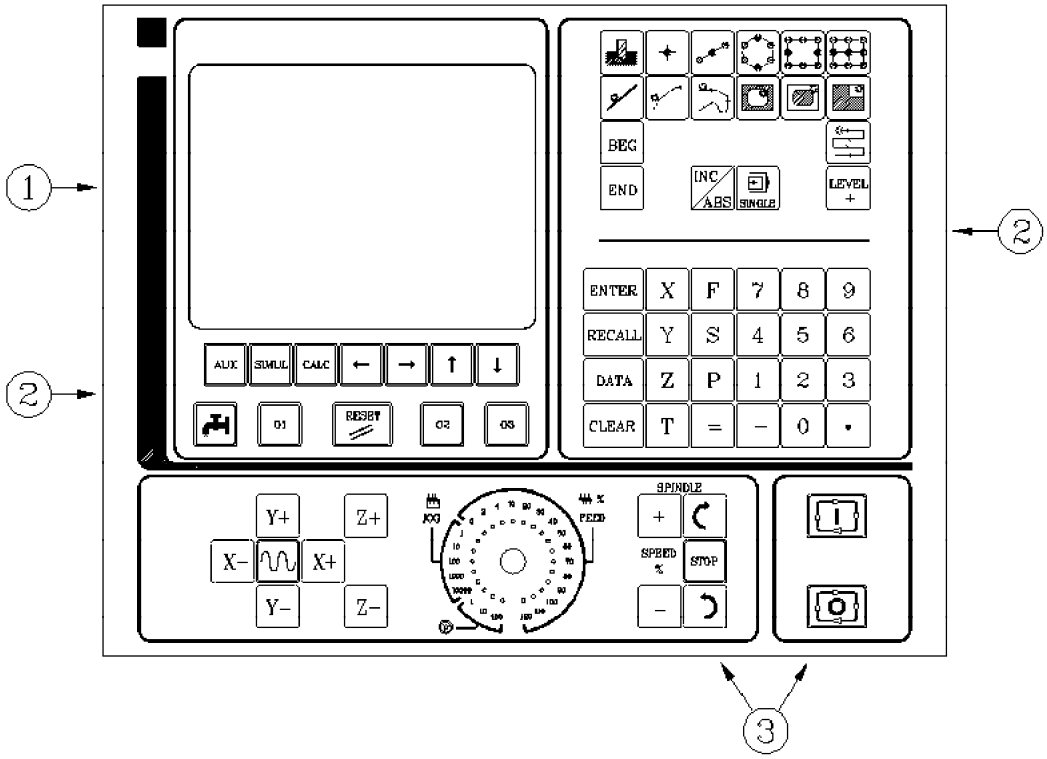
Atention:

The CNC is prepared to be used in Industrial Environments, especially on milling machines.

It can control machine movements and devices.

1.1 INTRODUCTION

The 800M CNC is an enclosed compact module whose front view offers:



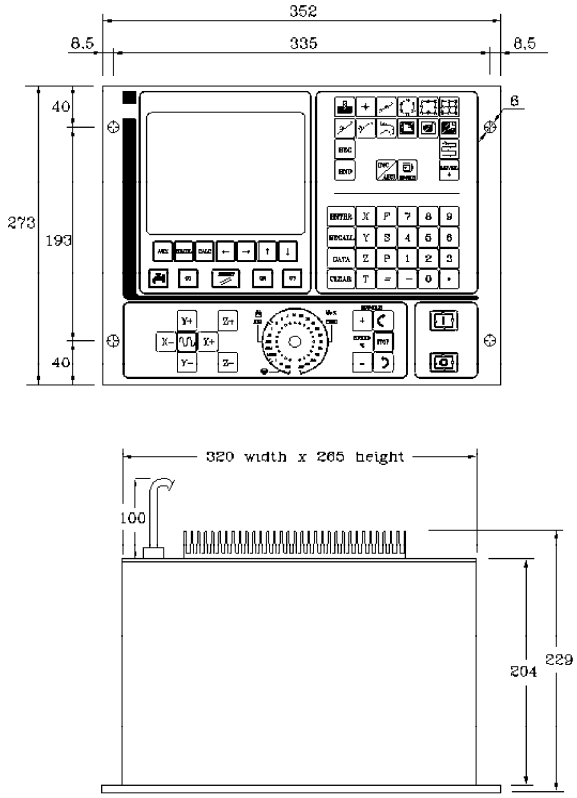
1. An 8" monochrome amber monitor or CRT screen used to display the required system information.

2. A keyboard which permits communications with the CNC; being possible to request information or change the CNC status by generating new instructions.

3. An operator panel containing the necessary keys to work in JOG mode as well as the Cycle Start/Stop keys.

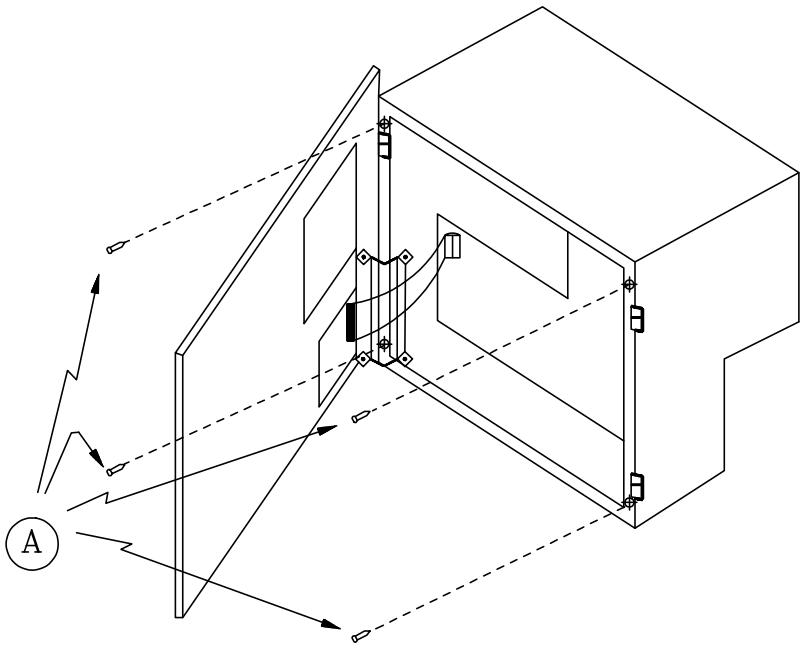
1.2 DIMENSIONS AND INSTALLATION

This CNC, usually mounted on the machine pendant, has 4 mounting holes.

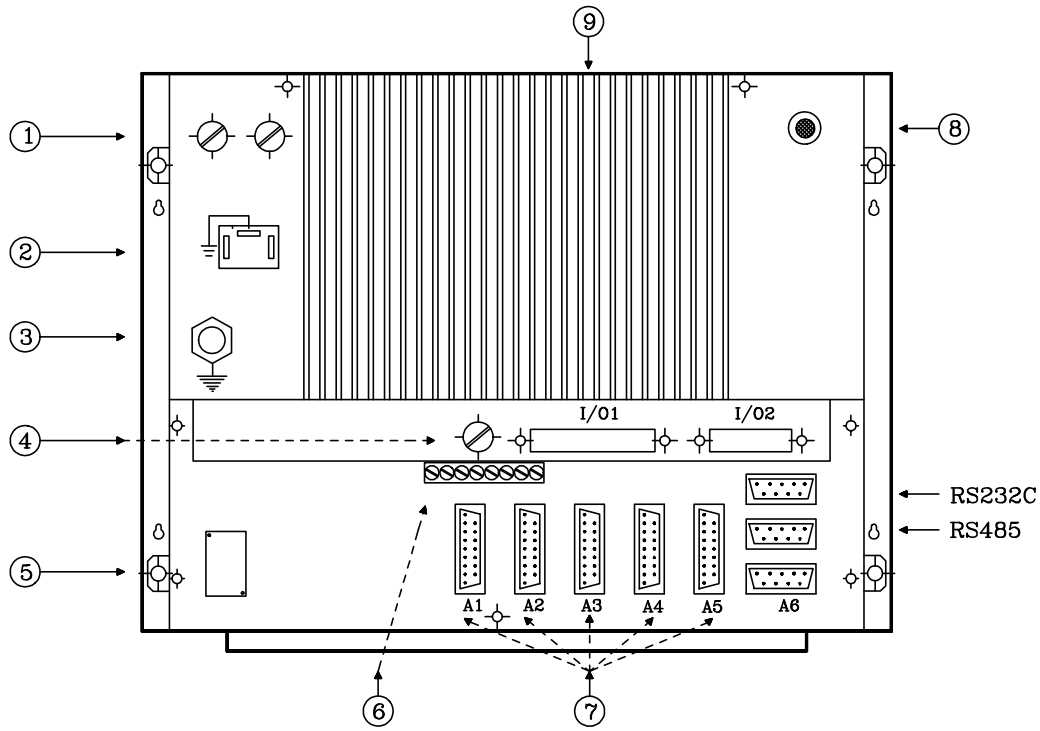


When installing it, leave enough room to swing the FRONT PANEL open in order to allow future access to its interior.

To open it, undo the 4 allen-screws located next to the CNC mounting holes.



1.3 CONNECTORS AND INTERFACE



A1 **15-pin SUB-D type female connector** to connect the X axis feedback system. It accepts sine-wave signal.

A2 **15-pin SUB-D type female connector** to connect the Y axis feedback system. It accepts sine-wave signal.

A3 **15-pin SUB-D type female connector** to connect the Z axis feedback system. It accepts sine-wave signal.

A4 **15-pin SUB-D type female connector** to connect the handwheel associated with the Y axis. It accepts sine-wave signal.

A5 **15-pin SUB-D type female connector** to connect the handwheel associated with the Z axis. It **does not** accept sine-wave signal.

A6 **9-pin SUB-D type female connector** to connect the handwheel associated with the X axis. It **does not** accept sine-wave signal.

RS485 **9-pin SUB-D type female connector.** Not being used at this time.

RS232C **9-pin SUB-D type female connector** to connect the RS232C serial line.

I/O1 **37-pin SUB-D type female connector** to interface with the electrical cabinet.

I/O2 **25-pin SUB-D type female connector** to interface with the electrical cabinet.

1- **Main AC fuse**. It has two 3.15Amp./250V. fast fuses (F), one per AC line, to protect the main AC input.

2- **AC power connector** To power the CNC. It must be connected to the power transformer and to ground.

3- **Ground terminal**. It must be connected to the general machine ground point. Metric 6.

4- **Fuse.** 3.15Amp./250V fast fuse (F) to protect the internal I/O circuitry of the CNC.

5- **Lithium battery.** Maintains the RAM data when the system's power disappears.

6- **Adjustment potentiometers for the analog outputs**. ONLY TO BE USED BY THE TECHNICAL SERVICE DEPARTMENT.

7- **10 dip-switches**. There are 2 under each feedback connector (A1 thru A5) and they are utilized to set the CNC according to the type of feedback signal being used.

8 **CRT brightness adjustment potentiometer**

9 **Heat-sink**.

Atention:

Do not manipulate the connectors with the unit connected to main AC power

Before manipulating these connectors (inputs/outputs, feedbach, etc) make sure that the unit is not connected to main AC power

1.3.1 CONNECTORS A1, A2, A3, A4

They are 15-pin SUB-D type female connectors used to connect the feedback signals.

* Connector A1 for X axis feedback signals.
* Connector A2 for Y axis feedback signals.
* Connector A3 for Z axis feedback signals.
* Connector A4 for Y axis electronic handwheel feedback signals.

The cable must have overall shield. The rest of the specifications depend on the feedback system utilized and the cable length required.

It is highly recommended to run these cables as far as possible from the power cables of the machine.

PIN	SIGNAL AND FUNCTION	
1 2 3 4	$\overline{A}$ A $\overline{B}$ B	Differential square-wave feedback signals
5 6	$\overline{Io}$ Io	Machine Reference Signals (marker pulses)
7 8	Ac Bc	Sine-wave feedback signals
9 10 11 12 13 14	+5V. 0V. -5V. 	Power to feedback system. Not connected. Power to feedback system. Not connected. Power to feedback system. Not connected.
15	CHASSIS	Shield

Atention:

When using square-wave rotary encoders, their signals must be TTL compatible. Encoders with open collector outputs MUST NOT be used.

Do not manipulate the connectors with the unit connected to main AC power

Before manipulating these connectors (inputs/outputs, feedbach, etc) make sure that the unit is not connected to main AC power

1.3.1.1 DIP-SWITCHES FOR CONNECTORS A1, A2, A3, A4

There are 2 dip-switches below each feedback input connector (A1 thru A4) to set the CNC according to the type of feedback signal being used.

Switch 1 indicates whether the feedback signal is sine-wave or square-wave and switch 2 indicates whether the feedback signal is single- or double-ended (differential).

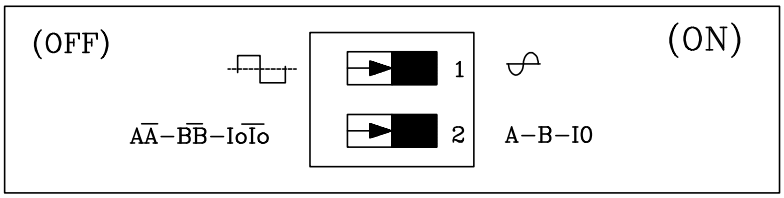
The possible types of feedback signals to be used at connectors A1 thru A4 are:

* Sine-wave (Ac, Bc, Io)
* Single-ended square-wave (A, B, Io)
* Double-ended (differential) square-wave (A, $\overline{A}$, B, $\overline{B}$, Io, $\overline{Io}$)

To select the type of signal for each axis, use the switch combinations below:

Dip-switch 1	Dip-switch 2	SIGNAL AND FUNCTION
ON	ON	Single-ended sine-wave signal (Ac,Bc,Io)
ON	OFF	Double-ended sine-wave signal **"Not allowed"**
OFF	ON	Single-ended square-wave signal (A,B,Io)
OFF	OFF	Double-ended square-wave (A, $\overline{A}$, B, $\overline{B}$, Io, $\overline{Io}$)

There is a label next to each dip-switch pair indicating the meaning of each switch.



1.3.2 CONNECTOR A5

It is a 15-pin SUB-D type female connector to connect the electronic handwheel to the Z axis. **It does not accept sine-wave signals.**

When using the spindle encoder and an electronic handwheel, the CNC will only control up to 4 axes. This connector will then be used for the spindle encoder or the electronic handwheel (the other device will be connected to A6).

The cable must have overall shield. The rest of the specifications depend on the feedback system utilized and the cable length required.

It is highly recommended to run these cables as far as possible from the power cables of the machine.

PIN	SIGNAL AND FUNCTION	
1 2 3 4	$\overline{A}$ $\overline{A}$ $\overline{B}$ $\overline{B}$	Double-ended square-wave signal.
5 6 7 8		Not being used at this time Not being used at this time Not being used at this time Not being used at this time
9 10 11 12 13 14	+5V. 0V. -5V.	Power to feedback system. Not connected. Power to feedback system. Not connected. Power to feedback system. Not connected.
15	CHASSIS	Shield.

Atention:

When the handwheel outputs square-wave signals, they must be TTL compatible. Handwheel with open collector outputs MUST NOT be used.

Do not manipulate the connectors with the unit connected to main AC power

Before manipulating these connectors (inputs/outputs, feedbach, etc) make sure that the unit is not connected to main AC power

1.3.2.1 DIP-SWITCHES FOR CONNECTOR A5

There are 2 dip-switches below this feedback input connector to set the CNC according to the type of feedback signal being used.

Switch 1 indicates whether the feedback signal is sine-wave or square-wave and switch 2 indicates whether the feedback signal is single- or double-ended (differential).

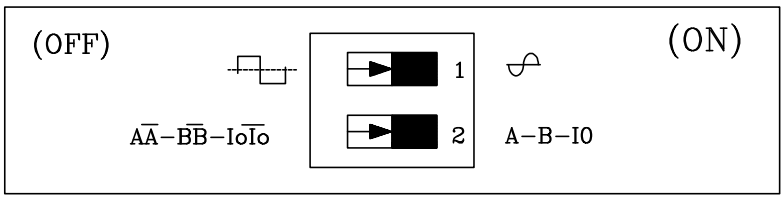
The possible types of feedback signals to be used at connector A5 are:

* Single-ended square-wave (A, B, Io)
* Double-ended (differential) square-wave (A, $\overline{A}$, B, $\overline{B}$, Io, $\overline{Io}$)

To select the type of signal for each axis, use the switch combinations below:

Dip-switch 1	Dip-switch 2	SIGNAL AND FUNCTION
ON	ON	Single-ended sine-wave signal **"Not allowed"**
ON	OFF	Double-ended sine-wave signal **"Not allowed"**
OFF	ON	Single-ended square-wave signal (A,B,Io)
OFF	OFF	Double-ended square-wave (A, $\overline{A}$, B, $\overline{B}$, Io, $\overline{Io}$)

There is a label next to each dip-switch pair indicating the meaning of each switch.



1.3.3 CONNECTOR A6

It is a 9-pin SUB-D type female connector to connect the electronic handwheel associated with the X axis. **It does not take sine-wave signals.**

The cable must have overall shield. The rest of the specifications depend on the feedback system utilized and the cable length required.

It is highly recommended to run these cables as far away as possible from the power cables of the machine.

PIN	SIGNAL AND FUNCTION	
1 2	A B	Square-wave signals from the spindle encoder or from the electronic handwheel
3		Axis selector signal (FAGOR 100P)
4 5	+5V 0V	Supply voltage for electronic handwheel
6 7 8		Not being used at this time Not being used at this time Not being used at this time
9	CHASSIS	Shield.

Atention:

When the handwheel outputs square-wave signals, they must be TTL compatible. Handwheel with open collector outputs MUST NOT be used.

When using the FAGOR 100P handwheel, the axis selector signal must be connected to pin 3.

Do not manipulate the connectors with the unit connected to main AC power

Before manipulating these connectors (inputs/outputs, feedbach, etc) make sure that the unit is not connected to main AC power

1.3.4 RS232C CONNECTOR

9-pin SUB-D type female connector to connect the RS 232 C serial port.

The cable shield must be soldered to pin 1 at the CNC end and to the metallic housing at the peripheral end.

PIN	SIGNAL	FUNCTION
1	FG	Shield
2	TxD	Transmit Data
3	RxD	Receive Data
4	RTS	Request To Send
5	CTS	Clear To Send
6	DSR	Data Send Ready
7	GND	Ground
8	—-	Not connected
9	DTR	Data Terminal Ready

SUGGESTIONS FOR THE RS232C INTERFACE

* **Connect/disconnect peripheral.** The CNC must be powered off when connecting or disconnecting any peripheral through this connector.

* **Cable length**. EIA RS232C standards specify that the capacitance of the cable must not exceed 2500pF; therefore, since average cables have a capacitance between 130pF and 170pF per meter, the maximum length of the cable should not be greater than **15m (49ft).**

  For greater distances, it is suggested to intercalate RS232C-to-RS422A signal converters (and vice-versa). Contact the corresponding distributor.

  Shielded cable with twisted-pair wires should be used to avoid communication interference when using long cables.

  Use shielded 7-conductor cable of 7*0.14mm² section.

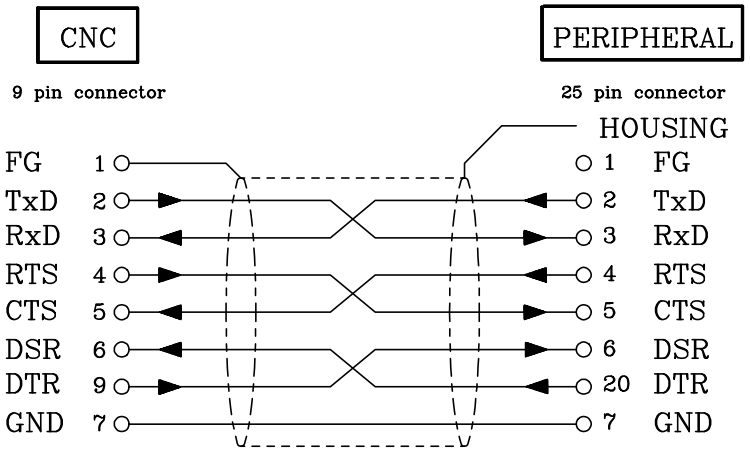
* **Transmission speed (baudrate)**. The baudrate normally used with peripherals is 9600 baud.

  All unused wires should be grounded to avoid erroneous control and data signals.

* **Ground connection**. It is suggested to reference all control and data signals to the same ground cable (pin 7 GND) thus, avoiding reference points at different voltages especially in long cables.

RECOMMENDED CONNECTIONS FOR THE RS232C INTERFACE

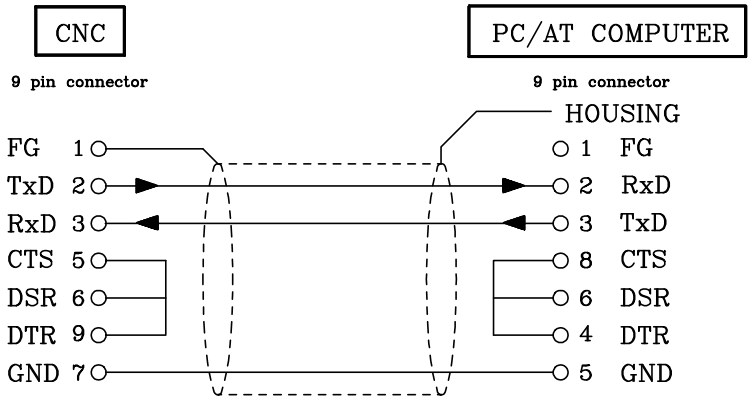
* Complete connection

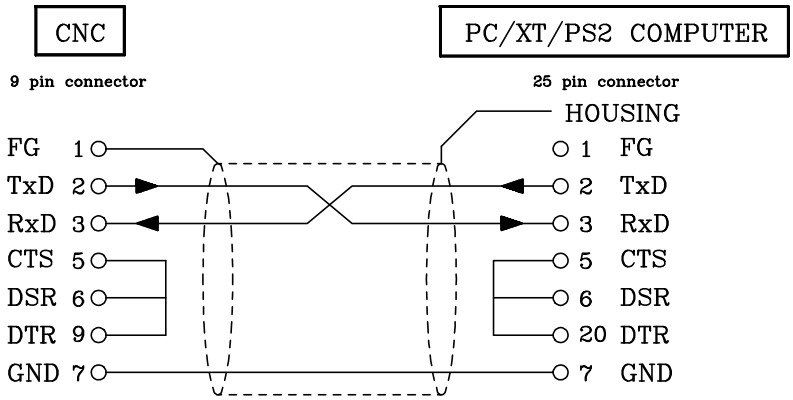


* Simplified connection

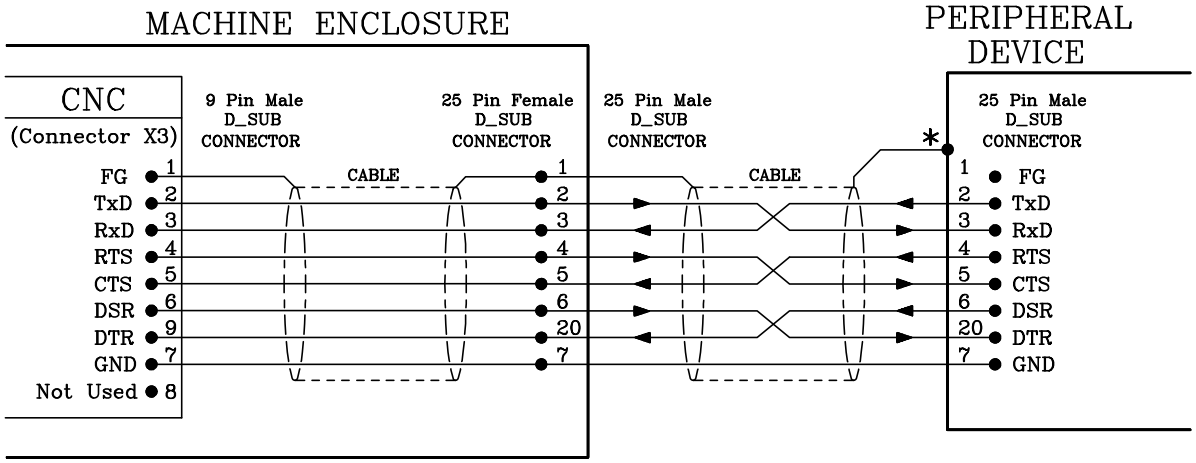
To be used when the peripheral or the computer meets one of the following requirements:

- It does not have the RTS signal.
- It is connected via DNC.
- The receiver can receive data at the selected baudrate.

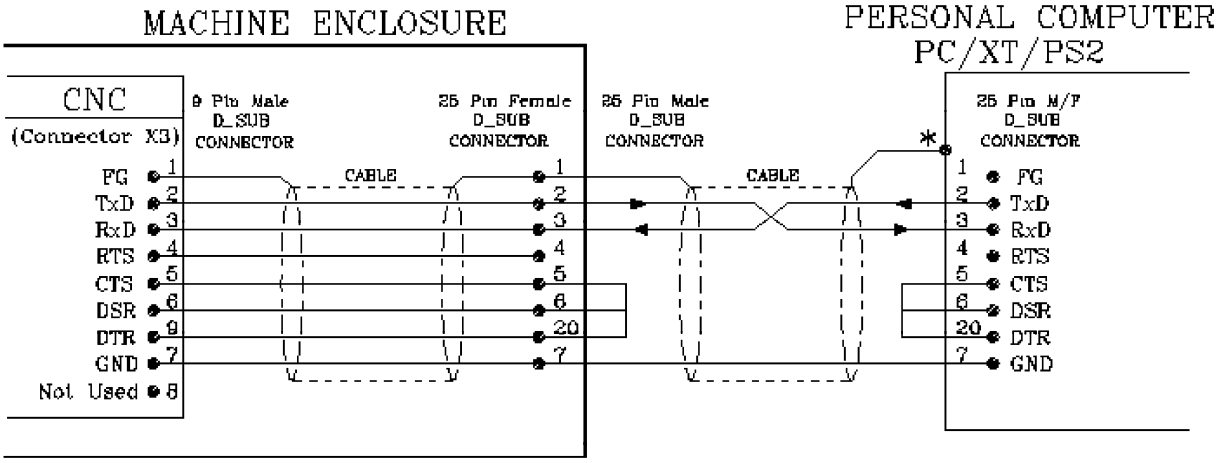




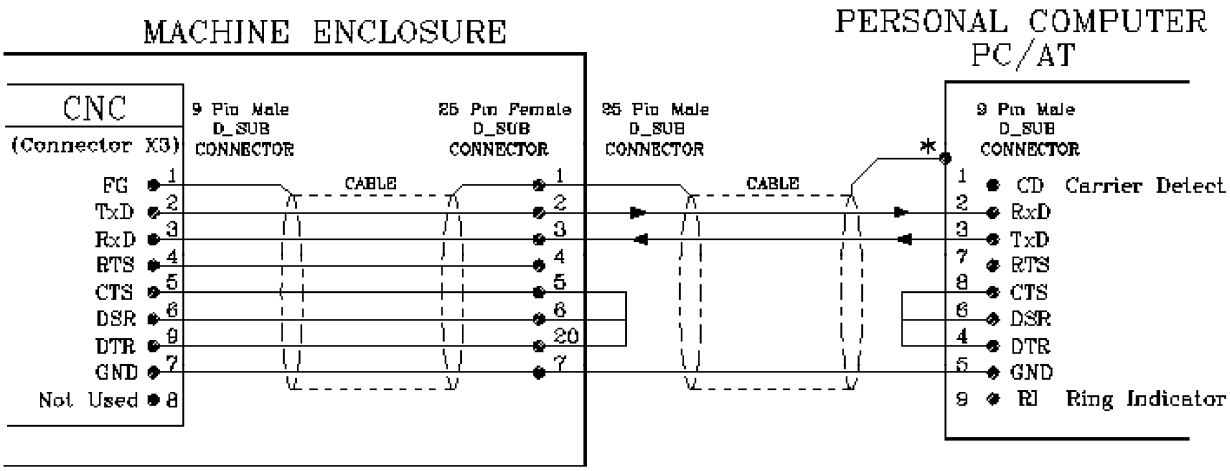
Nevertheless, it is suggested to refer to the technical manuals of the peripheral equipment in case there should be any discrepancy.



NOTE: VERIFY THE GENDER OF THE SERIAL PORT CONNECTOR USED ON YOUR PERIPHERAL DEVICE BEFORE MAKING THE CABLE ASSEMBLY



NOTE: VERIFY THE GENDER OF THE SERIAL PORT CONNECTOR USED ON YOUR PERSONAL COMPUTER BEFORE MAKING THE CABLE ASSEMBLY



* It is recommended to connect the cable shield to the Peripheral device or Personal Computer chassis in order to improve transmissions

1.3.5 CONNECTOR I/O 1

It is a 37-pin SUB-D type female connector to interface with the electrical cabinet.

Pin	SIGNAL AND FUNCTION	
1	0V.	Input from external power supply
2	T Strobe	Output. The BCD outputs represent a tool code.
3	S Strobe	Output. The BCD outputs represent a spindle speed code.
4	M Strobe	Output. The BCD outputs represent an M code.
5	Emergency	Output.
6		*Not being used at this time.*
7	Z Enable	Output.
8	Y Enable	Output.
9	X Enable	Output.
10	X home switch	Input from machine reference switch.
11	Y home switch	Input from machine reference switch.
12	Z home switch	Input from machine reference switch.
13		*Not being used at this time.*
14	$\overline{\text{Emergency}}$ Stop	Input.
15	Feed Hold $\overline{\text{Transfer}}$ inhibit $\overline{\text{M-done}}$	Input.
16	$\overline{\text{Stop}}$	Input.
17	Start Rapid JOG	Input
18	Block Skip	Conditional Input
19	DRO	Input. The CNC acts as a DRO
20	MST80	BCD coded output, weight: 80
21	MST40	BCD coded output, weight: 40
22	MST20	BCD coded output, weight: 20
23	MST10	BCD coded output, weight: 10
24	MST08	BCD coded output, weight: 8
25	MST04	BCD coded output, weight: 4
26	MST02	BCD coded output, weight: 2
27	MST01	BCD coded output, weight: 1
28	CHASSIS	Connect all cable shields to this pin.
29	24V.	Input from external power supply.
30	±10V	Analog output for X axis servo drive.
31	0V.	Analog output for X axis servo drive.
32	±10V	Analog output for Y axis servo drive.
33	0V.	Analog output for Y axis servo drive.
34	±10V	Analog output for Z axis servo drive.
35	0V.	Analog output for Z axis servo drive.
36	±10V	Analog output for the spindle drive.
37	0V.	Analog output for the spindle drive.

Atention:

The machine manufacturer must comply with the EN 60204-1 (IEC-204-1) regulation regarding the protection against electrical shock derived from defective input/output connection with the external power supply when this connector is not connected before turning the power supply on.

Do not manipulate the connectors with the unit connected to main AC power

Before manipulating these connectors (inputs/outputs, feedbach, etc) make sure that the unit is not connected to main AC power

1.3.5.1 INPUTS OF CONNECTOR I/O 1

0 V. Pin 1

INPUT from external power supply.

X AXIS HOME SWITCH Pin 10

This INPUT must be high (24V) as long as the machine reference switch for the X axis is pressed.

Y AXIS HOME SWITCH Pin 11

This INPUT must be high (24V) as long as the machine reference switch for the Y axis is pressed.

Z AXIS HOME SWITCH Pin 12

This INPUT must be high (24V) as long as the machine reference switch for the Z axis is pressed.

EMERGENCY STOP Pin 14

This INPUT must be normally high (24V).

When set low (0V), the CNC deactivates the axis enables and analog voltages, it interrupts the part program execution and it displays ERROR 64 on the CRT.

It **does not** imply an emergency output (pin 5 of this connector).

FEED HOLD / TRANSFER INHIBIT / M-DONE Pin 15

This INPUT must be normally high (24V) and its meaning depends on the type of block or function being executed at the time.

* If while moving the axes this signal (FEED-HOLD) is set low (0V), the CNC maintains the spindle turning and stops the axes bringing their analog voltages to 0V while maintaining their enables active.

  When this input is brought back high (24V), the axes will resume their movements.

* If while executing a motionless block this signal (TRANSFER INHIBIT) is set low (0V), the CNC interrupts the program execution at the end of the block currently in execution.

  When this signal is brought back high, the CNC resumes program execution.

* The "M-DONE" signal is used when machine parameter P605(5) is set to "1".

  The CNC waits for the electrical cabinet to execute the requested miscellaneous M function. In other words, it waits for the "M-done" input to be set high (24V).

STOP Pin 16

This INPUT must be normally high (24V)

When this input is set low (0V), the CNC interrupts the program execution just as if the key were pressed at the OPERATOR PANEL.

To resume program execution, it is necessary to bring this input back high (24V) and press the key at the OPERATOR PANEL.

START / RAPID JOG Pin 17

This INPUT must be normally low (0V).

* If an up-flank (leading edge or low-to-high transition) of this signal (START) is detected, the CNC considers that the external CYCLE START key is pressed and it behaves as if the key were pressed at the OPERATOR PANEL.

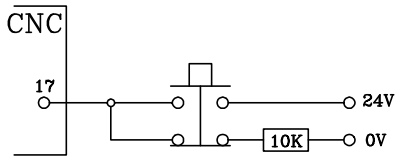
  However, to disable the key of the OPERATOR PANEL in order to only use this input, set machine parameter P618(1) to "1".

* When machine parameter P609(7) has been set to "1" and this input (RAPID JOG) is high (24V), the CNC acts as if the key were pressed.

  The CNC will perform all G01, G02 and G03 movements at 200% of the programmed feedrate F. If the resulting feedrate is greater than the maximum established by machine parameter P708, the CNC will issue the corresponding error message.

  Also, in the JOG mode and while this input is maintained high (24V), all movements will be carried out in rapid (G00).

While inactive, this input must be connected to 0V through a 10KOhm resistor.



BLOCK SKIP (Conditional input) Pin 18

Every time the CNC executes the miscellaneous function M01 (conditional stop), it analyzes the status of this input. If high (24V), the CNC will interrupt the execution of the program.

By the same token, every time the CNC must execute a conditional block, it will analyze the status of this input and it will execute the block if this input is high (24V).

DRO (DRO mode) Pin 19

If this input is set high (24V) while in the JOG mode, the CNC acts as a DRO.

1.3.5.2 OUTPUTS OF CONNECTOR I/O 1

T Strobe Pin 2

The CNC sets this output high (24V) whenever it sends a tool code (T function) via the BCD outputs (pins 20 thru 27).

S Strobe Pin 3

The CNC sets this output high (24V) whenever it sends a spindle speed code (S function) via the BCD outputs (pins 20 thru 27).

M Strobe Pin 4

The CNC sets this output high (24V) whenever it sends an M function code via the BCD outputs (pins 20 thru 27).

EMERGENCY Pin 5

The CNC activates this output whenever it detects an alarm condition or internal emergency.

This output is normally high (24V) or low (0V) depending on the setting of machine parameter P605(8).

Z AXIS ENABLE Pin 7

The CNC sets this output high (24V) to enable the Z axis servo drive.

Y AXIS ENABLE Pin 8

The CNC sets this output high (24V) to enable the Y axis servo drive.

X AXIS ENABLE Pin 9

The CNC sets this output high (24V) to enable the X axis servo drive.

MST80 Pin 20
MST40 Pin 21
MST20 Pin 22
MST10 Pin 23
MST08 Pin 24
MST04 Pin 25
MST02 Pin 26
MST01 Pin 27

The CNC uses these outputs to indicate to the electrical cabinet the M, S or T function that has been selected.

This information is BCD coded and the significance (weight) of each output is expressed by the corresponding mnemonic.

For example, to select the first spindle speed range, the CNC sends the M41 code out to the electrical cabinet.

MST80	MST40	MST20	MST10	MST08	MST04	MST02	MST01
0	1	0	0	0	0	0	1

Together with these signals, the CNC will activate the "M Strobe", "T Strobe" or "S Strobe" output to indicate the type of function being selected.

CHASSIS Pin 28

This pin must be used to connect all cable shields to it.

Analog voltage for X ±10V. Pin 30
Analog voltage for X 0V. Pin 31

These outputs provide the analog voltage for the X axis servo drive. The cable used for this connection must be shielded.

Analog voltage for Y ±10V. Pin 32
Analog voltage for Y 0V. Pin 33

These outputs provide the analog voltage for the Y axis servo drive. The cable used for this connection must be shielded.

Analog voltage for Z ±10V. Pin 34
Analog voltage for Z 0V. Pin 35

These outputs provide the analog voltage for the Z axis servo drive. The cable used for this connection must be shielded.

Spindle analog voltage ±10V. Pin 36
Spindle analog voltage 0V. Pin 37

These outputs provide the analog voltage for the spindle drive. The cable used for this connection must be shielded.

1.3.6 CONNECTOR I/O 2

It is a 25-pin SUB-D type female connector to interface with the electrical cabinet.

PIN	SIGNAL AND FUNCTION	
1	0V.	Input from external power supply.
2	0V.	Input from external power supply.
3	Output M1 Coolant	Value of bit 1 of the decoded M function table.
4	Output M2	Value of bit 2 of the decoded M function table.
5	Output M3	Value of bit 3 of the decoded M function table.
6	Output M4	Value of bit 4 of the decoded M function table.
7	Output M5	Value of bit 5 of the decoded M function table.
8	Output M6	Value of bit 6 of the decoded M function table.
9	Output M7	Value of bit 7 of the decoded M function table.
10	Output M8	Value of bit 8 of the decoded M function table.
11	Output M9	Value of bit 9 of the decoded M function table.
12	Output M10 Mandatory Io	Value of bit 10 of the decoded M function table.
13	Output M11	Value of bit 11 of the decoded M function table.
14		*Not being used at this time*
15		*Not being used at this time*
16	CHASSIS	Connect all cable shields to this pin.
17		*Not being used at this time*
18		*Not being used at this time*
19	24V.	Input from external power supply.
20	24V.	Input from external power supply.
21	JOG	Output. JOG mode is selected.
22	Output M15	Value of bit 15 of the decoded M function table.
23	Output M14 Reset	Value of bit 14 of the decoded M function table.
24	Output M13	Value of bit 13 of the decoded M function table. Indicates that a program block is being executed. Indicates that program P99996 is being executed. Indicates that the axes are being positioned in rapid.
25	Output M12 Vertical axis	Value of bit 12 of the decoded M function table.

Atention:

The machine manufacturer must comply with the EN 60204-1 (IEC-204-1) regulation regarding the protection against electrical shock derived from defective input/output connection with the external power supply when this connector is not connected before turning the power supply on.

Do not manipulate the connectors with the unit connected to main AC power

Before manipulating these connectors (inputs/outputs, feedbach, etc) make sure that the unit is not connected to main AC power

1.3.6.1 OUTPUTS OF CONNECTOR I/O 2

"Decoded M" outputs Pins 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 22, 23, 24, 25

These OUTPUTS provide the values indicated at the table corresponding to the selected M function.

For example: If the table corresponding to function M41 has been set as follows:

M41 100100100100100 (outputs to be activated)
00100100100100100 (outputs to be deactivated)

Every time this M41 function is executed, the CNC will act as follows:

	M01	M02	M03	M04	M05	M06	M07	M08	M09	M10	M11	M12	M13	M14	M15
Pin I/O2	3	4	5	6	7	8	9	10	11	12	13	25	24	23	22
at 24V	x			x			x			x			x		
at 0V			x			x			x			x			x
Not modified		x			x			x			x			x	

Outputs M01 / Coolant Pin 3

This output, besides providing the value of bit 1 of the decoded table corresponding to the selected M function, acts as the output for the coolant.

When using it as the coolant output, be careful not to set the bit of the decoded M table which corresponds to this M1 output since the CNC will activate it in both cases.

Nevertheless, the CNC maintains this output active whenever the coolant is active even when executing an M function which would deactivate it.

Outputs M10 / Mandatory Io Pin 12

This output provides the value of bit 10 of the decoded table corresponding to the selected M function.

If machine parameter P611(2) has been set to "1", the CNC sets this output high on power-up and keeps it high until all the axes are referenced (homed).

Care must be taken when using this option not use the bit of the decoded M table corresponding to this output M10 since the CNC will activate it in both cases.

JOG Pin 21

The CNC sets this OUTPUT high (24V) whenever the JOG mode is selected.

Outputs M14 / RESET Pin 23

This OUTPUT provides the value of bit 14 of the decoded M table corresponding to the selected M function.

If machine parameter P609(3) is set to "1" to provide a RESET pulse, this positive reset pulse will be output every time the CNC executes a RESET.

Care must be taken, when having this option, not to use the bit of the decoded M table corresponding to this output M14 since the CNC will activate it in both cases.

Outputs M13 / Program block in execution / P99996 in execution / Rapid positioning in progress Pin 24

This OUTPUT provides the value of bit 13 of the decoded M table corresponding to the selected M function.

If machine parameter P611(1) is set to "1", the CNC sets this OUTPUT high (24V) every time a part-program block is being executed.

If machine parameter P611(6) is set to "1", the CNC sets this OUTPUT high (24V) whenever the special ISO-coded user program P99996 is being executed.

If machine parameter P613(4) is set to "1", the CNC sets this OUTPUT high (24V) whenever the CNC is executing a rapid positioning move

Care must be taken, when having one of these options, not to use the bit of the decoded M table corresponding to this output M13 since the CNC will activate it in all these cases.

Outputs M12 / Vertical axis movement Pin 25

This OUTPUT provides the value of bit 12 of the decoded M table corresponding to the selected M function.

If machine parameter P613(2) is set to "1" in order for the CNC to provide the status of the vertical axis movement, this output will indicate the direction of that movement. If the axis is moving in the positive direction (counting up), this output will be set low (0V) and it will be set high (24V) if moving in the negative direction (counting down).

Care must be taken, when having this option, not to use the bit of the decoded M table corresponding to this output M12 since the CNC will activate it in both cases.

2. *POWER AND MACHINE INTERFACE*

Atention:

Power switch
This power switch must be mounted in such a way that it is esaily accessed and at a distance between 0.7 meters (27.5 inches) and 1.7 meters (5.5 ft) off the floor.

Intall this unit in the proper place
It is recommended to install the CNC away from coolants, chemical products, possible blows etc. which could damage it.

2.1 POWER INTERFACE

The rear of the 800M CNC has a three-prong connector for AC and ground connection.

This connection must be done through an independent shielded 110VA transformer with an AC output voltage between 100V and 240V +10% -15%.

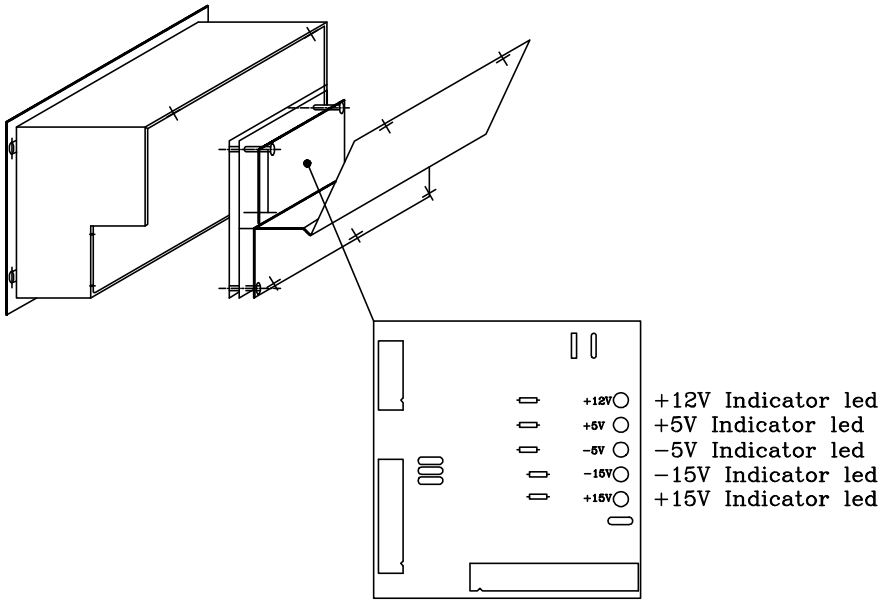
The power outlet for the equipment must be near it and with easy access to it.

In case of overload or overvoltage, it is recommended to wait for 3 minutes before powering the unit back up in order to prevent any possible damage to the power supply.

2.1.1 INTERNAL POWER SUPPLY

Inside the 800M CNC there is a power supply providing the required voltages.

Besides the 2 outside AC power fuses (one per line), it has a 5 Amp. fuse inside to protect it against overcurrent.



2.2 MACHINE INTERFACE

2.2.1 GENERAL CONSIDERATIONS

The machine tool must have decoupled all those elements capable of generating interference (relay coils, contactors, motors, etc.).

* D.C. Relay coils.

  Diode type 1N4000.

* A.C. relay coils

  RC connected as close as possible to the coils. Their approximate values should be:

  R 220 Ohms/1W
  C 0,2 µF/600V

* A.C. motors.

  RC connected between phases with values:

  R 300 Ohms/6W
  C 0,47µF/600V

Ground connection.

It is imperative to carry out a proper ground connection in order to achieve:

* Protection of anybody against electrical shocks caused by a malfunction.

* Protection of the electronic equipment against interference generated by the proper machine or by other electronic equipment near by which could cause erratic equipment behavior.

Therefore, it is crucial to install one or two ground points where the above mentioned elements must be connected.

Use large section cables for this purpose in order to obtain low impedance and efficiently avoid any interference. This way, all parts of the installation will have the same voltage reference.

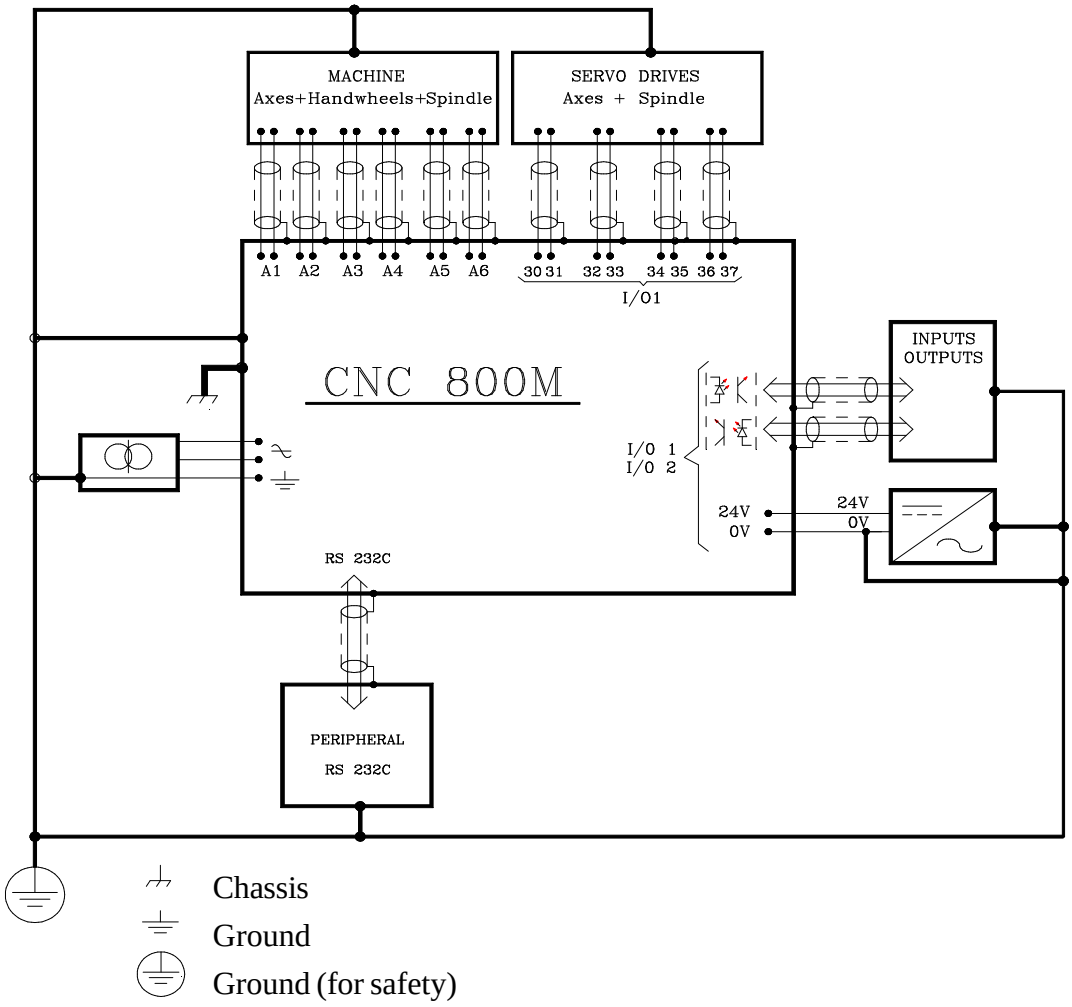
Even when a proper **ground** connection reduces the effects of electrical interference (noise), the signal cables require additional protection.
This is generally achieved by using twisted-pair cables which are also covered with anti-static shielding mesh-wire. This shield must be connected to a specific point avoiding **ground loops** that could cause undesired effects. This connection is usually done at one of the CNC's ground points.

Each element of the machine-tool/CNC interface must be connected to ground via the established main points. These points will be conveniently set close to the machine-tool and properly connected to the general ground (of the building).

When a second point is necessary, it is recommended to join both points with a cable whose section is not smaller than 8 mm².

Verify that the impedance between the central point of each connector housing and the main ground point is less than 1 Ohm.

Ground connection diagram



2.2.2 DIGITAL OUTPUTS.

The CNC has several optocoupled digital outputs which may be used to activate relays, deacons, etc.

These digital outputs, with galvanic isolation by optocouplers, can commutate D.C. voltages supplied by the electrical cabinet of the machine.

The electrical characteristics of these outputs are:

Nominal voltage value	+24 V D.C.
Maximum voltage value	+30 V D.C.
Minimum voltage value	+18 V D.C.
Output voltage	Vcc.- 2V
Maximum output current	100 mA.

All outputs are protected by means of:

Galvanic isolation by optocouplers.
External 3A fuse for protection against output overload (greater than 125mA), external power supply overvoltage (over 33V DC) and against reverse connection of the external power supply.

2.2.3 DIGITAL INPUTS.

The digital inputs of the CNC are to used to "read" external devices.

All of them are galvanically isolated from the outside world by optocouplers.

The electrical characteristics of these inputs are:

Nominal voltage value	+24 V DC
Maximum voltage value	+30 V.
Minimum voltage value	+18 V.
High threshold voltage (logic state 1) over	+18V.
Low threshold voltage (logic state 0) under	+5V.
Typical input consumption	5 mA.
Maximum consumption per input	7 mA.

All inputs are protected by means of:

Galvanic isolation by optocouplers.
Protection against reverse connection of the power supply up to -30V.

Atention:

The external 24V power supply used for the digital inputs and outputs must be regulated.

The 0V point of this power supply must be connected to the main ground point of the electrical cabinet.

2.2.4 ANALOG OUTPUTS.

The CNC has 6 analog outputs which could be used to command servo drives, spindle drives and other devices.

The electrical characteristics of these outputs are:

Analog voltage range:	±10V.
Minimum impedance of the connected drive:	10 KOhm.
Maximum cable length without shield:	75 mm.

It is highly recommended to use the shielded cable connecting the shield to the corresponding pin of the connector.

Atention:

It is recommended to adjust the servo drives so their maximum feedrate (G00) is obtained at ±9.5 V.

2.2.5 FEEDBACK INPUTS

The feedback inputs are used to receive sine-wave, single-ended and double-ended square-wave signals coming from linear or rotary transducers (encoders).

Connector A1 is used for the X axis feedback signals and it accepts sine-wave and double-ended (differential) square-wave signals.

Connector A2 is used for the Y axis feedback signals and it accepts sine-wave and double-ended (differential) square-wave signals.

Connector A3 is used for the Z axis feedback signals and it accepts sine-wave and double-ended (differential) square-wave signals.

Connector A4 is used for the handwheel associated to the Y axis and it accepts sine-wave and double-ended (differential) square-wave signals.

Connector A5 is used for the handwheel associated to the Z axis and it accepts double-ended (differential) square-wave signals.

Connector A6 is used for the handwheel associated to the X axis and it accepts single-ended (not differential) square-wave signals.

The electrical characteristics of these inputs are:

Sine-wave signals	Supply voltage	±5V.±5%
	Maximum counting frequency	25KHz.
Square-wave signals	Supply voltage	±5V.±5%
	Maximum counting frequency	200KHz.

It is recommended to use shielded cables for their connection connecting the shield to the corresponding pin of the connector.

2.3 SET-UP

2.3.1 GENERAL CONSIDERATIONS

Inspect the whole electrical cabinet verifying the ground connections BEFORE powering it up.

This ground connection must be done at a single machine point (Main Ground Point) and all other ground points must be connected to this point.

Verify that the 24V external power supply used for the digital inputs and outputs is REGULATED and that its 0V are connected to the Main Ground Point.

Verify the connection of the feedback system cables to the CNC.

DO NOT connect or disconnect these cables to/from the CNC when the CNC is on.

Look for short-circuits in all connectors (inputs, outputs, axes, feedback, etc.) BEFORE supplying power to them.

2.3.2 PRECAUTIONS

It is recommended to reduce the axis travel installing the limit switches closer to each other or detaching the motor from the axis until they are under control.

Verify that there is no power going from the servo drives to the motors.

Verify that the connectors for the digital inputs and outputs are disconnected.

Verify that the feedback dip-switches for each axis are set according to the type of feedback signal being used.

Verify that the E-STOP button is pressed.

2.3.3 CONNECTION

Verify that the AC power is correct.

Being the CNC disconnected, power the electrical cabinet and verify that it responds properly.

Verify that there is proper voltage between the pins corresponding to 0V and 24V of the connectors for the digital inputs and outputs.

Apply 24V to each one of the terminals of the electrical cabinet being used that correspond to the digital outputs of the CNC and verify their correct performance.

With the motors being decoupled from the axes, verify that the system consisting of drive, motor and tacho is operating properly.

Connect the AC power to the CNC. After a self-test, the CNC will show the message: "GENERAL TEST PASSED". If there is any problem, the CNC will display the corresponding error message.

2.3.4 SYSTEM INPUT/OUTPUT TEST

This CNC offers a work mode which allows the possibility to activate or deactivate each one of the logic inputs and outputs of the CNC.

To do this, press the following keystroke sequence:

[AUX]	(SPECIAL FUNCTIONS)
[5]	(AUXILIARY MODES)
[1]	(SPECIAL MODES)
[0] [1] [0] [1]	(Access code) (password)
[0]	(TEST)

Once the CNC has performed the system self-test, press **[7]** and the CNC will show the status of the logic inputs being possible to change the status of the logic outputs.

Logic inputs

INPUT	PIN	FUNCTION
A	17 (I/O 1)	START
B	16 (I/O 1)	STOP
C	15 (I/O 1)	FEEDHOLD
D	14 (I/O 1)	EMERGENCY STOP
E		To be used only by the technical service
F	12 (I/O 1)	Z axis home switch
G	11 (I/O 1)	Y axis home switch
H	10 (I/O 1)	X axis home switch
I	19 (I/O 1)	DRO mode
J	18 (I/O 1)	Block skip (conditional stop)
K		To be used only by the technical service
L		To be used only by the technical service
M		To be used only by the technical service
N		To be used only by the technical service

The CNC will show at all times and dynamically the status of all these inputs. To check a specific one, just actuate on the external push-button or switch observing its behavior on the CRT.

The value of "1" on the screen indicates that the corresponding input is receiving 24V DC and a "0" indicates that it doesn't.

Logic outputs

OUTPUT	ROW 1 PIN/FUNCTION	ROW 2 PIN/FUNCTION
A	(2 I/O 1) T Strobe	(3 I/O 2) Output 1, decoded M
B	(3 I/O 1) S Strobe	(4 I/O 2) Output 2, decoded M
D	(5 I/O 1) Emergency	(6 I/O 2) Output 4, decoded M
E	(6 I/O 1) Threading ON	(7 I/O 2) Output 5, decoded M
F	(7 I/O 1) Z Enable	(8 I/O 2) Output 6, decoded M
G	(8 I/O 1) Y Enable	(9 I/O 2) Output 7, decoded M
H	(9 I/O 1) X Enable	(10 I/O 2) Output 8, decoded M
I	(27 I/O 1) MST01	(11 I/O 2) Output 9, decoded M
J	(26 I/O 1) MST02	(12 I/O 2) Output 10, decoded M
K	(25 I/O 1) MST04	(13 I/O 2) Output 11, decoded M
L	(24 I/O 1) MST08	(25 I/O 2) Output 12, decoded M
M	(23 I/O 1) MST10	(24 I/O 2) Output 13, decoded M
N	(22 I/O 1) MST20	(23 I/O 2) Output 14, decoded M
O	(21 I/O 1) MST40	(22 I/O 2) Output 15, decoded M
P	(20 I/O 1) MST80	(21 I/O 2) CNC in JOG mode

To check one of these outputs, select it with the cursor which may be moved by means of the right and left arrow keys.

Once the desired output is selected, press "1" to activate it and "0" to deactivate it. The CRT will show the status change.

It is possible to have several outputs active at the same time providing 24V at their corresponding pins.

Once the INPUT/OUTPUT test is completed, disconnect the electrical cabinet and, then, connect the input/output connectors as well as the feedback systems of the axes to the CNC.

Then, connect the electrical cabinet and the CNC to AC power and activate the servo drives.

Nevertheless, to quit the I/O testing mode, press **[END].**

2.4 EMERGENCY INPUT/OUTPUT CONNECTION

The Emergency Input of the CNC is called EMERGENCY STOP (E-STOP) and corresponds to pin 14 of connector I/O1. This input must normally have 24V DC.

The CNC processes this signal directly, therefore, whenever these 24V disappear, it will issue EXTERNAL EMERGENCY ERROR (Error 64), it will deactivate the axes enables and cancel the analog voltages for all the axes and the spindle.
It does **NOT** imply the emergency output (pin5).

The electrical cabinet interface must take into account all the external elements that could cause this error.

For example, some of these elements may be:

* The E-Stop button has been pressed.

* An axis travel limit switch has been pressed.

* An axis servo drive is not ready.

On the other hand, whenever a CNC detects an internal emergency error, it will activate the EMERGENCY OUTPUT at pin 5 of connector I/O1.

This output will be normally high or low depending on the setting of machine parameter P605(8).

There are some of the internal causes that can activate this output:

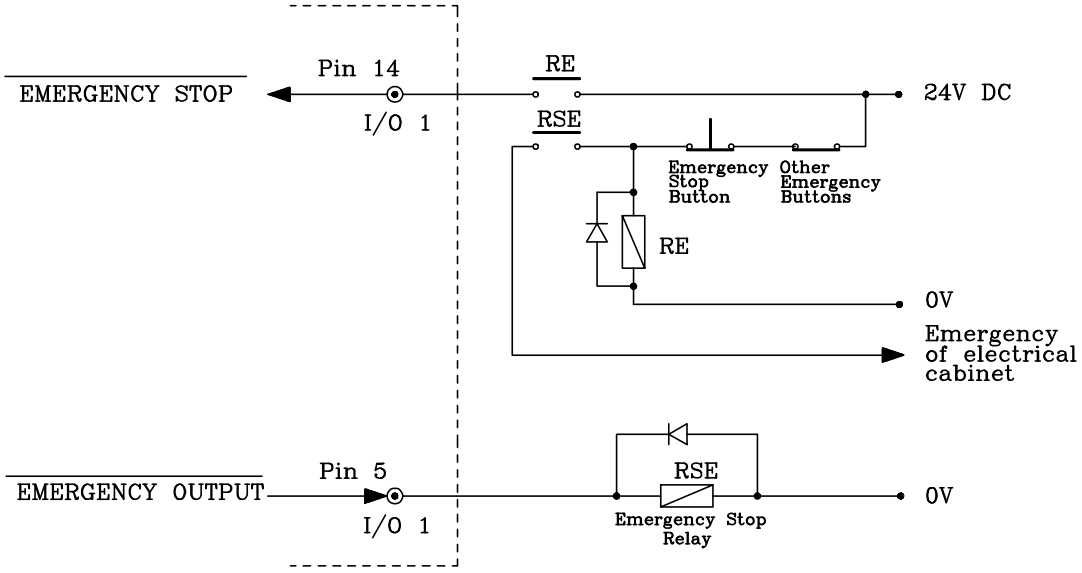
* An excessive axis following error has occurred.

* An axis feedback error has occurred.

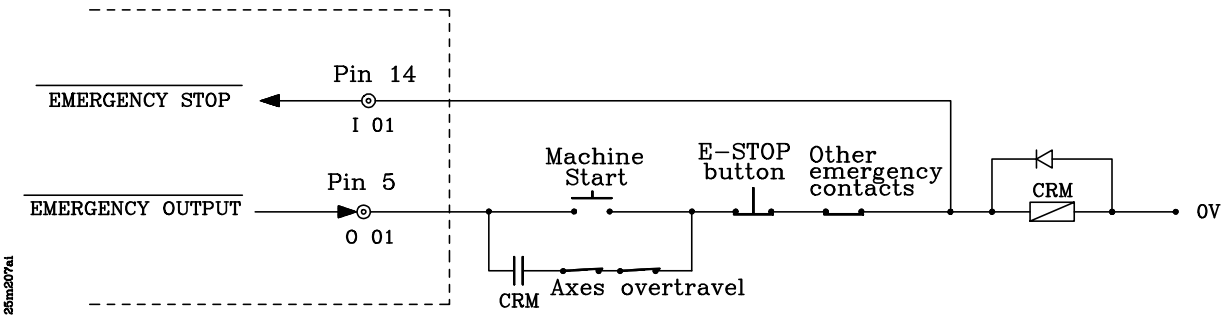
* There is erroneous data on the machine parameter table.

The recommended connection when P605(8)= 1 (output normally HIGH) is:

European Style:

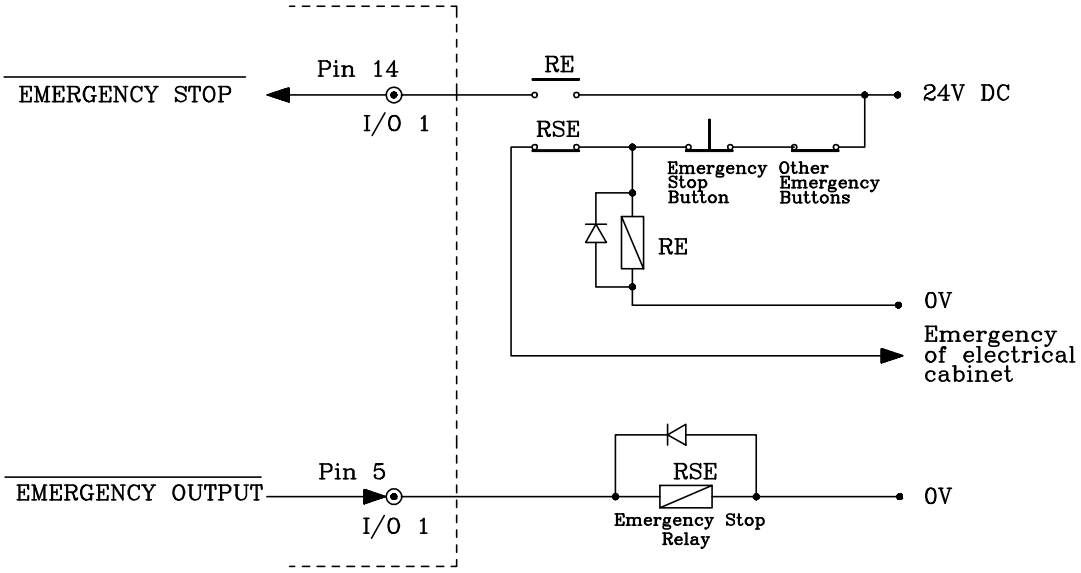


USA Style:

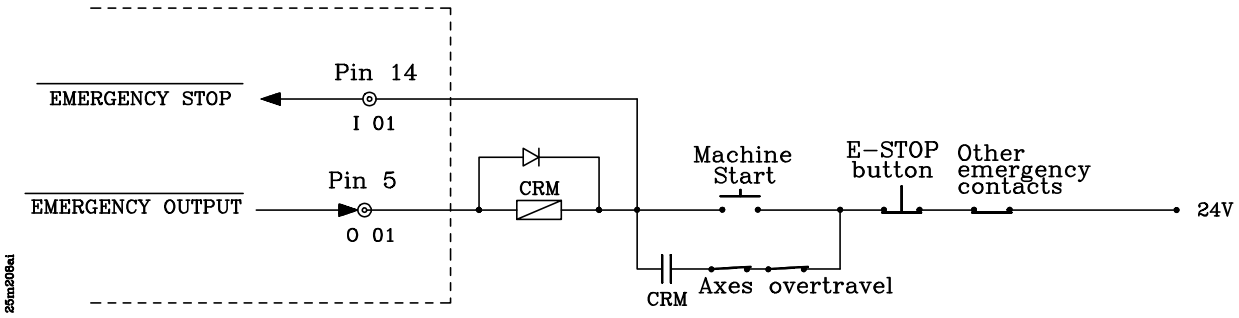


The recommended connection when P605(8)= 0 (output normally LOW) is:

European Style:



USA Style:



2.5 ACTIVATION / DEACTIVATION OF EXTERNAL DEVICES

With this CNC it is possible to activate and deactivate up to 4 external devices including the coolant. The other devices depend on the type of machine available.

To do this, the following keys are available:

If it is selected (lamp on); pin 3 of connector I/O2 stays high (24V).

When it is not selected, (lamp off); pin 3 of connector I/O2 stays low (0V)

When this device (O1) is selected, the CNC sends the M10 function out to activate it and an M11 to deactivate it.

When this device (O2) is selected, the CNC sends the M12 function out to activate it and an M13 to deactivate it.

When this device (O3) is selected, the CNC sends the M14 function out to activate it and an M15 to deactivate it.

The coolant may be activated or deactivated at any time but the other devices (O1, O2 and O3) must be activated or deactivated only when the axes of the machine are in position.

3. AUXILIARY FUNCTIONS

Press **[AUX]** to access this option.

The CNC will show a series of options. To select one of them, simply press the number key indicated for the desired option.

The operator may access all the shown options except the one referred to as "AUXILIARY MODES". When selecting this option, the CNC requests the password to access the various tables and operating modes exclusive for the OEM.

Press **[END]** to quit any of these options and return to the standard display mode.

3.1 MILLIMETERS <—> INCHES

When selecting this option, the CNC changes the display units from millimeters to inches and vice versa and it displays the X, Y, Z coordinates in the selected units.

Also, The axes feedrates are also shown in the new selected units.

It must be borne in mind that the values stored in BEGIN, END as well as the data for special operations and the coordinates corresponding to the "point to point movements" have no units. Therefore, their values will remain unchanged when shifting from mm to inches and vice versa.

3.2 TOOL LENGTH COMPENSATION

Every time this option is selected, the CNC activates or deactivates tool length compensation.

When **not** operating with **tool length** compensation, the CNC displays the coordinates of the **tool base**.

When operating with tool length compensation, machine parameter P626(1) determines whether the CNC is to display the coordinate of the tool tip or that of the tool base.

The right-hand side of the screen shows the symbol: whenever tool length compensation is active.

3.3. TOOL TABLE

When selecting this option, the CNC shows the values assigned to each tool offset, that is, the dimensions of each tool being used to machine the parts.

Once the tool offset table has been selected, the operator will be able to move the cursor over the screen one line at a time by means of the up and down arrow keys.

Each tool offset has a series of fields defining the tool dimensions. These fields are:

R ***Tool radius.***

It will be given in the currently selected work units, its maximum value being:

R 1000.000 mm or R 39.3700 inches.

The CNC will apply this value when tool radius compensation is active.

L ***Tool length.***

It will be given in the currently selected work units, its maximum value being:

L 1000.000 mm or L 39.3700 inches.

The CNC will apply this value when tool length compensation is active.

I ***Tool radius wear compensation.***

It will be given in the currently selected work units, its value range being:

I ±32.766 mm or I ±1.2900 inches.

The CNC will add this value to the nominal radius value "R" to calculate the real radius (R+I).

K ***Tool length wear.***

It is given in the currently selected work units, its value range being:

K ±32.766 mm or K ±1.2900 inches

The CNC will add this value to the nominal tool length value "L" to calculate the real (total) tool length (L+K).

3.3.1 MODIFICATION OF TOOL DIMENSIONS

To clear the whole tool table by setting all its fields to 0, key in the following keystroke sequence: **[F] [S] [P] [ENTER]**.

The 800M CNC has the "TOOL CALIBRATION" option described next. Once the tools have been calibrated, the CNC assigns to each tool offset the dimensions of the corresponding tool.

To modify the table values of a tool ("R", "L", "I" and "K"), first, select, at the CNC, the corresponding tool offset by keying in the desired tool number and pressing **[RECALL]**.

The CNC will show at the editing area the values currently assigned to that tool offset.

To modify these values, move the pointer with the up and down arrow keys until it is located on the current value. The new values must be keyed in over those currently assigned.

Once the new values have been keyed in, press **[ENTER]** so they are stored in memory.

To quit this mode, move the pointer to the right until it is out of the editing area and, then, press **[END]**.

3.4 TOOL CALIBRATION

With this option it is possible to calibrate and load the dimensions of the tools onto the tool offset table of the CNC.

The CNC shows, at the lower right-hand side of the screen, a graphic aide to guide the operator during the tool calibration process and it will highlight the data requested at each moment

The tool calibration process consists of the following steps:

1.- The CNC requests the known Z axis dimension of the work-piece (contact point used for calibration).

Key in this value and press **[ENTER]**.

2.- The CNC requests the number of the tool to be calibrated (T).

Key in the desired tool number and then press **[ENTER]**.

3.- Move the machine with either the mechanical handwheels, the electronic handwheel or the jog keys until the tool tip touches the part along the Z axis.

Then, press **[ENTER]**. The CNC will calibrate the tool in length and it will update the corresponding tool offset.

The CNC will then request a new tool to be calibrated. Repeat steps 2 and 3 for each new tool to be added.

Press **[END]** to quit this mode and return to the standard display mode.

Atention:

During tool calibration, it is possible to use the electronic handwheels, the jog keys of the operator panel as well as the spindle control keys of the operator panel.

3.5 EXECUTION / SIMULATION OF PROGRAM P99996

Program P99996 is a special user program in ISO code. It must be edited (written) on a PC and sent to the CNC via the Peripherals option.

When selecting the option: "Program P99996 Execution", This program may be either executed by pressing [I] or simulated by pressing [SIMUL]

3.5.1 EXECUTION OF PROGRAM P99996

When selecting the option: "Execution of program P99996", the CNC displays the following information:

```
AUTOMATIC        P99996   N0000
N00 G90
N10 G94
N20 T1.1
N30 F2000

     COMMAND       ACTUAL        TO GO

   X 0000.000    X 0000.000    X 0000.000
   Y 0000.000    Y 0000.000    Y 0000.000
   Z 0000.000    Z 0000.000    Z 0000.000

F0000.000 %100   S0000 %100   T00.00
G05 01 95
M41
```

The top line shows the message "AUTOMATIC", the program number (P99996) and the number of the first block of the program or that of the block being in execution.

Then, the CRT shows the contents of the first program blocks. If the program is being executed, the first block of the list will be the one being executed at the time.

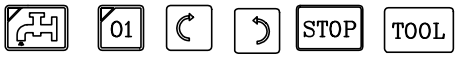
The position values along X, Y and Z indicate the programmed values (COMMAND), the current position (ACTUAL) and the distance remaining (TO GO) for the axes to reach the "command" position.

It also shows the selected spindle speed, programmed value multiplied by the active %S override (COMMAND), and the real spindle speed (ACTUAL).

The bottom of the screen shows the machining conditions currently selected. The programmed feedrate F, the % F override, the programmed spindle speed S, the %S override, the programmed Tool as well as the active G and M functions.

To execute program P99996, press [I] . It always starts executing from the first block.

To interrupt the program, press [O] . Once interrupted, the following keys are enabled:



To resume execution, press [I]

3.5.1.1 TOOL INSPECTION

With this option it is possible to interrupt the execution of program P99996 and inspect the tool to check its status and change it if necessary.

To do this, follow these steps:

a) Press to interrupt the program.

b) Press [**T**]

At this time, the CNC executes the miscellaneous function M05 to stop the spindle and it displays the following message on the screen:

JOG KEYS AVAILABLE
OUT

c) Move the tool to the desired position by using the JOG keys.

Once the tool is "out of the way", the spindle may be started and stopped again by its corresponding keys at the Operator Panel.

d) Once the tool inspection or replacement is completed, press [**END**].

The CNC will execute an M03 or M04 function to start the spindle in the direction it was turning when the program was interrupted.

The screen will display the following message:

RETURN
AXES OUT OF POSITION

"Axes out of position" means that they are not at the position where the program was interrupted.

e) Jog the axes to the program interruption position by means the corresponding jog keys. The CNC will not allow to move them passed (overtravel) this position.

When the axes are in position, the screen will display:

RETURN
AXES OUT OF POSITION
NONE

f) Press to resume the execution of program P99996.

3.5.1.2 EXECUTION MODES

With this CNC, it is possible to execute program P99996 from beginning to end without interruptions or block by block by pressing [SINGLE]

The top line of the screen shows the selected operating mode: "Automatic" or "Single Block".

To switch modes, press [SINGLE] again.

Once the desired execution mode has been selected, press [I] to run the program.

3.5.1.3 CNC RESET

This option is used to reset the CNC setting it to the initial conditions established by the machine parameters. When quitting this operating mode, the CNC displays the DRO mode.

To reset the CNC, interrupt the program if it is running and simply press [RESET]

The CNC will request comfirmation of this function by blinking the message: **"RESET?"**.

To go ahead with reset, press [RESET] again; but to cancel it, press [CLEAR]

3.5.1.4 DISPLAYING PROGRAM BLOCKS

To display the previous or following blocks to those appearing on the screen, press:

[⇦] Displays the previous blocks

[⇨] Displays the following blocks

Atention:

Bear in mind that P99996 always starts executing from the first block of the program, regardless of the blocks currently displayed on the screen.

3.5.1.5 DISPLAY MODES

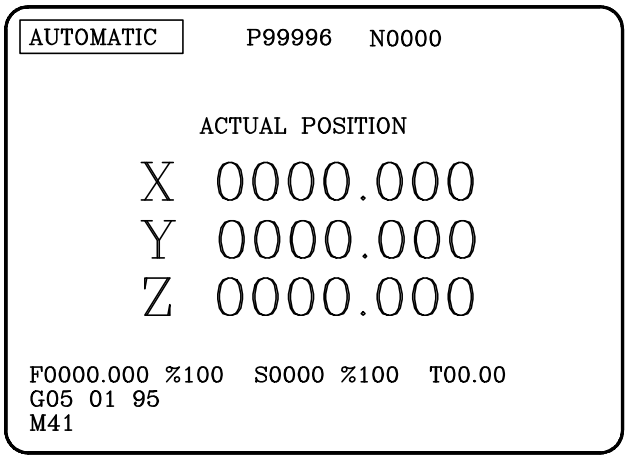
There are 4 display modes which can be selected by means of the following keys:

[0] STANDARD
[1] ACTUAL POSITION
[2] FOLLOWING ERROR
[3] ARITHMETIC PARAMETER

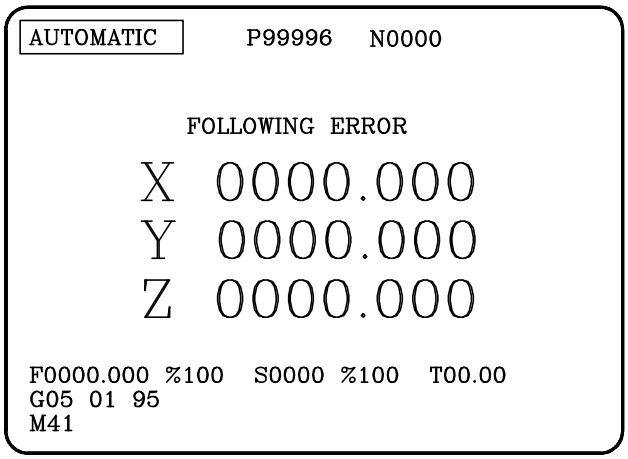
STANDARD display mode

It is the mode described before. When accessing the "Execution of program P99996" option, the CNC selects this display mode.

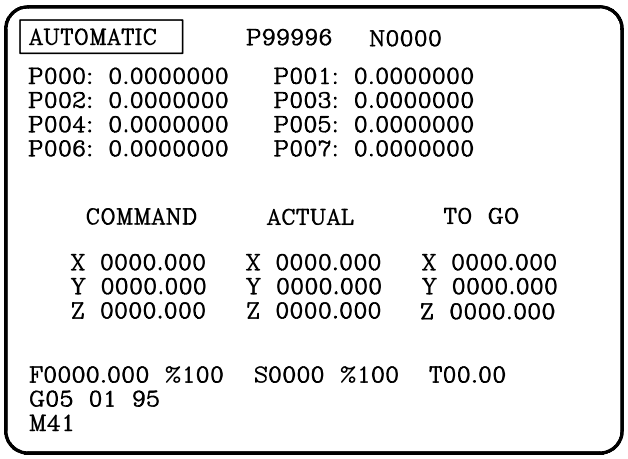
ACTUAL POSITION display mode



FOLLOWING ERROR display mode



ARITHMETIC PARAMETERS display mode



This mode shows a group of 8 arithmetic parameters. To view the previous and following ones, use these keys:

- Displays the previous parameters
- Displays the following ones

The value of each parameter may be expresssed in one of the following formats:

P46 = -1724.9281	Decimal notation
P47 = -.10842021 E-2	Scientific notation

Where "E-2" means 10^{-2} (1/100). Therefore, the two types of notation for the same parameter below have the same value:

P47= -0.001234	P47= -0.1234 E-2
P48= 1234.5678	P48= 1.2345678 E3

3.5.2 SIMULATION OF PROGRAM 99996

With the 800M CNC, it is possible to check program 99996 in dry run before executing it.

To do this, press [SIMUL]. The CNC will display a graphic screen.

The lower left-hand side of the screen shows the plane being represented and it is possible to chose between the XY, XZ, YZ planes or the 3-D (XYZ).

For the CNC to show another plane, press the corresponding key:

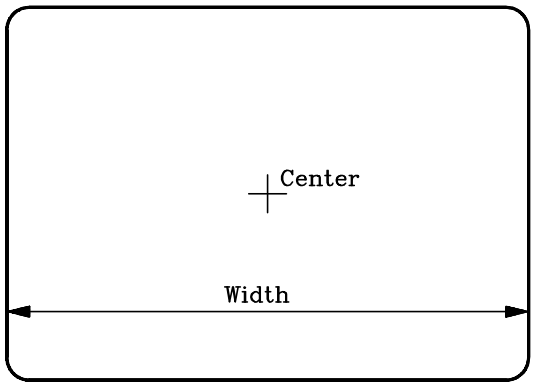
[0] XY Plane
[1] XZ Plane
[2] YZ Plane
[3] 3-D (XYZ)

This CNC may show the graphic representation in up to 3 planes; therefore, it will only show those currently selected. To select other planes, proceed as follows:

Press [SIMUL] , the CNC will ask whether each one of the possible planes is to be selected or not.

If wished to select the presented plane, press **[Y]** and if not, press **[ENTER]**.

Once the desired planes have been indicated, select the display area by indicating the XYZ coordinates of the point appearing at the center of the screen and the width of the display area. Press **[ENTER]** after keying in each value.



To check the part, press [I] and the corresponding graphic representation will begin.

While simulating, any other graphic plane is available (keys: 1, 2 and 3); but, no planes or display areas may be selected. To select planes or modify display areas, the program simulation must be interrupted by pressing [ ].

To delete the graphic, press **[CLEAR]** and to quit the simulation mode, press **[END]**.

3.5.2.1 ZOOM FUNCTION

With the ZOOM function, it is possible to enlarge or shrink either the whole graphic or part of it. To do this, the simulation of the program must be interrupted or ended.

Once the representation plane to be enlarged or shrunk has been selected, press **[Z]**. A rectangle will appear over the original graphic which represents the zoom window.

To alter the dimensions of this zoom window, use these keys:

[-] Reduces the size of the window (greater enlargement).

[=] Increases the size of the window (smaller enlargement).

To move the zoom window around, use: [◄] [►] [▲] [▼]

When the desired window size and location has been selected, press **[ENTER]**.

To see the selected zoom area enlarged while keeping the current display-area values, press [SIMUL]

The section of the graphic enclosed in the zoom window will now occupy the whole screen (zoom in).

To return to the previous whole area display (zoom out), press **[END]**.

To zoom in again, just press **[Z]** and proceed as before.

To quit the ZOOM mode and return to the regular graphic representation, press **[END]**.

3.6 AUXILIARY MODES

When selecting this option, the CNC shows the following menu:

1 - SPECIAL MODES
2 - PERIPHERALS
3 - LOCK / UNLOCK

After accessing one of these modes and operate with it, press **[END]** to quit. At this point, the CNC will show this menu again. Press **[END]** once more to return to the standard display mode.

3.7 SPECIAL MODES

When selecting this option, the CNC will request the password to access these auxiliary modes. This password is the following:

0101

Once this code has been entered, the CNC displays the following menu:

0 - TEST
1 - GENERAL PARAMETERS
2 - DECODED M FUNCTIONS
3 - LEADSCREW ERROR COMPENSATION

3.7.1 TEST

To select this option, press **[AUX]**, select the "Special Modes" option of the "Auxiliary Modes" menu, key in the password (**0101**) and press the key corresponding to "TEST".

The CNC performs the General Test.

Once the test is completed, it is possible to test the logic inputs and outputs of the CNC, verify the checksum corresponding to the software version currently installed or perform the general test of the CNC again.

*** Testing the logic inputs and outputs of the CNC.**

Press **[7]** to access this option, the CNC will display the status of the logic INPUTS and it is possible to simulate the logic OUTPUTS of the CNC.

The inputs indicated by the letters "A" thru "M" have the meaning shown by the chart below and their status is indicated by either a "0" or a "1".

A value of "0" means that it receives 0V
A value of "1" means that it receives 24V

	CNC LOGIC INPUTS	
	Meaning	*Pin*
A	Cycle Start	17 (I/O1)
B	Cycle Stop (It must be normally high)	16 (I/O1)
C	Feed Hold (It must be normally high)	15 (I/O1)
D	Emergency Stop (It must be normally high)	14 (I/O1)
E	Only to be used by the Technical Department	
F	Z axis home switch	12 (I/O1)
G	Y axis home switch	11 (I/O1)
H	X axis home switch	10 (I/O1)
I	Manual (DRO mode)	19 (I/O1)
J	Block Skip (conditional input)	18 (I/O1)
K	Only to be used by the Technical Department	
L	Only to be used by the Technical Department	
M	Only to be used by the Technical Department	
N	Only to be used by the Technical Department	

The logic outputs are shown in two rows under the letters "A" thru "M" and with the meanings indicated by the chart below.

Each output may be assigned a "0" or "1" value which means:

If "0", the corresponding output will be at 0V (low).
If "1", the corresponding output will be at 24V (high).

Use the up and down arrow keys to move the cursor and select the desired outputs.

	TOP ROW		**BOTTOM ROW**	
	Meaning	*Pin*	*Meaning*	*Pin*
A	T Strobe	2 (I/O1)	Output 1, decoded M	3 (I/O2)
B	S Strobe	3 (I/O1)	Output 2, decoded M	4 (I/O2)
C	M Strobe	4 (I/O1)	Output 3, decoded M	5 (I/O2)
D	Emergency	5 (I/O1)	Output 4, decoded M	6 (I/O2)
E	Threading ON	6 (I/O1)	Output 5, decoded M	7 (I/O2)
F	Z axis Enable	7 (I/O1)	Output 6, decoded M	8 (I/O2)
G	Y axis Enable	8 (I/O1)	Output 7, decoded M	9 (I/O2)
H	X axis Enable	9 (I/O1)	Output 8, decoded M	10 (I/O2)
I	MST01	27 (I/O1)	Output 9, decoded M	11 (I/O2)
J	MST02	26 (I/O1)	Output 10, decoded M	12 (I/O2)
K	MST04	25 (I/O1)	Output 11, decoded M	13 (I/O2)
L	MST08	24 (I/O1)	Output 12, decoded M	25 (I/O2)
M	MST10	23 (I/O1)	Output 13, decoded M	24 (I/O2)
N	MST20	22 (I/O1)	Output 14, decoded M	23 (I/O2)
O	MST40	21 (I/O1)	Output 15, decoded M	22 (I/O2)
P	MST80	20 (I/O1)	CNC in JOG mode	21 (I/O2)

*** Checksum of the software version**

Press **[8]** to access this option. The CNC will display the checksum of each EPROM memory corresponding to the software version currently installed at the CNC.

*** New general test of the CNC**

Press **[9]** to access this mode. The CNC will perform the general self-test again.

After accessing one of these tests (inputs/outputs, checksum or general test), press **[END]** to return to the "AUXILIARY MODES" menu and press **[END]** again to return to the standard display mode.

3.7.2 GENERAL PARAMETERS

To select this option, press **[AUX]**, select the "Special Modes" option of the "Auxiliary Modes" menu, key in the password (**0101**) and press the key corresponding to "GENERAL PARAMETERS".

The CNC shows the machine parameter table.

The operator may view the following or previous pages by means of the up and down arrow keys.

To display a particular parameter, key in the desired parameter number and press **[RECALL]**. The CNC will then display the page corresponding to that parameter.

To EDIT a parameter, key in the desired parameter number, press **[=]** and then, key in the desired value.

Depending on the type of machine parameter selected, the following types of values may be assigned:

*	A number	P111 = 30000
*	A group of 8 bits	P602 = 00001111
*	A character	P105 = 1 (YES)

Once the machine parameter has been set, press **[ENTER]** for that value to be entered on the table.

If when pressing **[=]**, the parameter being edited disappears from the screen, it means that the machine parameters are protected and cannot be modified.

It must be borne in mind that once all the machine parameters are set, either the **[RESET]** key must be pressed or the CNC must be turned off and back on in order for these values to be assumed by the CNC.

To lock or unlock the access to the machine parameters, to the decoded M function table and to the leadscrew error compensation table, proceed as follows:

* Press **[AUX]** and after selecting the "Lock/Unlock" option of the of the "Auxiliary Modes" menu...

* Key in: "P1111" and press **[ENTER]** to lock the access or: "P0000" **[ENTER]** to unlock it.

When the access to the machine parameter table is locked, only those parameters related to the RS 232 serial communications line may be edited.

Remember that once the desired parameters have been edited, **[RESET]** must be pressed or the CNC must be turned off and back on in order for the CNC to assume the new values.

The meaning of each parameter as well as the proper way to define them is described in another chapter of this manual.

3.7.3 DECODED "M" FUNCTIONS

To lock or unlock the access to the machine parameters, to the decoded M function table and to the leadscrew error compensation table, proceed as follows:

* Press **[AUX]** and after selecting the "Lock/Unlock" option of the of the "Auxiliary Modes" menu...

* Key in: "P1111" and press **[ENTER]** to lock the access or: "P0000" **[ENTER]** to unlock it.

To select this option, press **[AUX]**, select the "Special Modes" option of the "Auxiliary Modes" menu, key in the password (**0101**) and press the key corresponding to "DECODED M FUNCTIONS".

The CNC will show the decoded M function table.

The operator may view the following or previous pages by means of the up and down arrow keys.

To view a particular M function, key in its number and press **[RECALL]**. The CNC will then show the page corresponding to that function.

To EDIT a parameter, key in the desired parameter number, press **[=]**, key in the desired value and, then, press **[ENTER]** in order for this value to be entered on the table.

When executing an "M" function, outputs M1 thru M15 of connector I/O 2 will be modified depending on the setting of the corresponding function.

Two rows of "1s" and "0s" will appear to the right of each "M" function. The top row has 15 characters and the bottom one 17.

The **top-row** characters have the following meaning:

0 Indicates the outputs which do not change when the M function is executed. They keep the previous status.
1 Indicates the outputs which are **activated** (set to 24V) when the M function is executed.

The first 15 characters (from the left) of the **bottom row** have the following meaning:

0 Indicates the outputs which do not change when the M function is executed. They keep the previous status.
1 Indicates the outputs which are **deactivated** (set to 0V) when the M function is executed.

For example: If the M41 table (first spindle speed range selection) has been set as follows:

```
M41  100100100100100        (Outputs being activated)
     00100100100100100      (Outputs being deactivated)
```

The CNC will behave like this every time M41 is executed:

	M01	M02	M03	M04	M05	M06	M07	M08	M09	M10	M11	M12	M13	M14	M15
Pin I/O2	3	4	5	6	7	8	9	10	11	12	13	25	24	23	22
at 24V	x			x			x			x			x		
at 0V			x			x			x			x			x
Not modified		x			x			x			x			x	

To activate the BCD outputs "MST01" thru "MST80" (pins 20 thru 27 of connector I/O 1) besides, the decoded ones, machine parameter "P606 bit 7" must be set to "0".

Bit 16 of the bottom row indicates whether the "M" function is executed at the beginning (0) of the block or at the end (1) once the programmed movements have been executed.

Bit 17 of the bottom row determines whether the CNC must wait for confirmation from the electrical cabinet indicating that the execution of the M function has been completed (M-done) or not before resuming the execution of the program.

This confirmation is carried out by means of the "M-DONE" input at pin 15 of connector I/O 1. This bit may be set as follows:

0 The CNC **waits** for the "M-DONE" confirmation signal from the electrical cabinet.

1 The CNC **does not wait** for the "M-DONE" confirmation signal from the electrical cabinet.

Up to 32 M functions may be set. All those empty M-table positions are indicated as **M??**.

Whenever a previously defined M function is redefined, the new setting replaces the old one.

3.7.3.1 M FUNCTIONS SENT OUT IN BCD

The CNC generates a series of M functions to indicate to the electrical cabinet that a certain event has taken place.

The CNC activates the BCD outputs corresponding to the generated "M" function (pins 20 thru 27 of connector I/O 1).

If besides activating these BCD outputs, the decoded outputs are also to be activated (pins 3 thru 13 and 22 thru 25 of connector I/O 2), the corresponding M functions must be defined at the decoded M-function table.

The CNC generates the following M-functions in BCD:

M00 At the end of the execution of each step of the selected operation while in "SINGLE" mode.

M03 When pressing the key to start the spindle clockwise.

M04 When pressing the key to start the spindle counter-clockwise.

M05 When pressing the key to stop the spindle.

M10 When pressing the key to turn on the external device O1.

M11 When pressing the key to turn off the external device O1.

M12 When pressing the key to turn on the external device O2.

M13 When pressing the key to turn off the external device O2.

M14 When pressing the key to turn on the external device O3.

M15 When pressing the key to turn off the external device O3.

M30 When pressing the RESET key of the CNC.

M41 When selecting the first spindle speed range.

M42 When selecting the second spindle speed range.

M43 When selecting the third spindle speed range.

M44 When selecting the fourth spindle speed range.

3.7.4 LEADSCREW ERROR COMPENSATION

To select this option, press **[AUX]**, select the "Special Modes" option of the "Auxiliary Modes" menu, key in the password (**0101**) and press the key corresponding to "LEADSCREW ERROR COMPENSATION".

The CNC will display the Leadscrew Error Compensation table.

The operator may view the following or previous pages by using the up and down arrow keys.

To view a particular parameter, key in its number and press **[RECALL]**. The CNC will show the page corresponding to that parameter.

To clear the table by setting all the parameters to 0, key in the following sequence: **[F] [S] [P] [ENTER].**

There are up 30 parameter pairs for each axis. Parameters P0 thru P59 for the X axis, P60 thru P119 for the Y axis and P120 thru P179 for the Z axis.

Each parameter pair of this table represents:

Even parameter The position of the error point on the leadscrew. This position is referred to Machine Reference Zero (home).

Value range: ±8388.607 millimeters
±330.2599 inches

Odd parameter The amount of leadscrew error at that point.

Value range: ±32.766 millimeters
±1.2900 inches

When defining the compensation points on the table, the following rules must be observed:

* The even parameters are ordered according to their position along the axis. The first pair of parameters (P0, P60, P120) must be set for the most negative (least positive) point of the axis to be compensated.

* If all 30 points of the table are not required, set the unused ones to 0.

* For those sections outside the compensation area, the CNC will apply the compensation defined for the nearest point.

* The Machine Reference Zero point (home) must be set with an error of 0.

* The maximum difference between the error values of two consecutive compensation points must be within: ±0.127 mm (±0.0050 inches)

* The inclination of the error graph between two consecutive points cannot be greater than 3%.

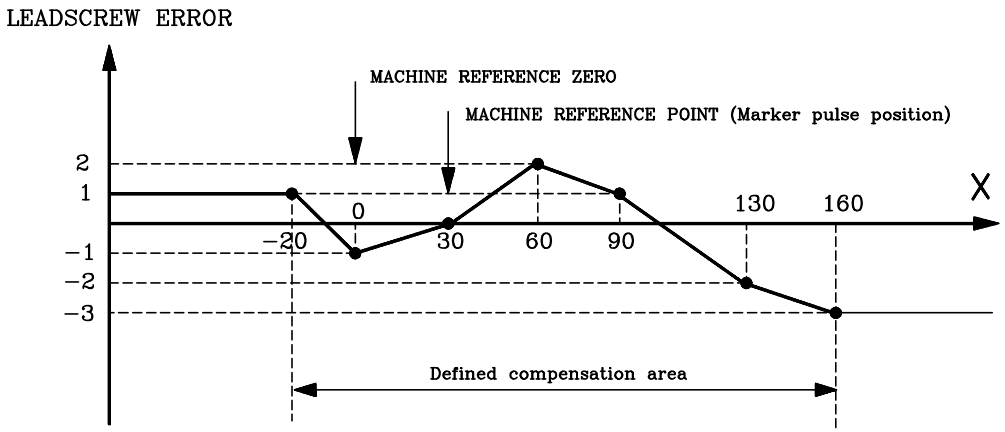
  Examples: If the distance between two consecutive points is 3 mm. the maximum difference of their relevant error values can be 0.090 mm.
  If the error difference between two consecutive points is the maximum (0.127mm), the distance between them cannot be smaller than 4.233mm.

To EDIT a parameter, key in its number, press **[=]**, key in the desired value and press **[ENTER]** so the new value is entered on the table.

Remember to press **[RESET]** or power the CNC off and back on once the machine parameters have been set in order for the CNC to assume their new values.

Programming example:

An X axis leadscrew is to be compensated according to the following graph in the section between X-20 and X160:



Considering that the machine reference **point** has a value of X30 (meaning that it is located 30mm from the Machine Reference **Zero**), the leadscrew error compensation parameters will be defined as follows:

P000	= X -20.000	P001	= X 0.001
P002	= X 0.000	P003	= X -0.001
P004	= X 30.000	P005	= X 0.000
P006	= X 60.000	P007	= X 0.002
P008	= X 90.000	P009	= X 0.001
P010	= X 130.000	P011	= X -0.002
P012	= X 160.000	P013	= X -0.003
P014	= X 0.000	P015	= X 0.000
P016	= X 0.000	P017	= X 0.000
"	"	"	"
"	"	"	"
P056	= X 0.000	P057	= X 0.000
P058	= X 0.000	P059	= X 0.000

3.8 PERIPHERALS

With this CNC it is possible to communicate with the FAGOR Floppy Disk Unit, with a general peripheral device or with a computer in order to transfer programs from and to one another. This communication may be managed either from the CNC when in the "**Peripheral mode**" or from the computer by means of FAGOR's DNC protocol in which case the CNC may be in any of its operating modes.

3.8.1 PERIPHERAL MODE

In this mode, the CNC may communicate with the FAGOR Floppy Disk Unit, with a general peripheral device or with a computer having a standard off-the-shelf communications program.

To access this mode, press **[AUX]** and select the "**Peripherals**" option of the "**Auxiliary modes**" menu.

The upper left-hand side of The CNC screen will show the following menu:

0 - RECEIVE FROM (Fagor) **FLOPPY DISK UNIT**
1 - SEND TO (Fagor) **FLOPPY DISK UNIT**
2 - RECEIVE FROM GENERAL DEVICE
3 - SEND TO GENERAL DEVICE
4 - (Fagor) **FLOPPY DISK UNIT DIRECTORY**
5 - (Fagor) **DELETE FLOPPY DISK UNIT PROGRAM**
6 - DNC ON/OFF

In order to use any of these options, the DNC mode must be **inactive.** If it is active (the upper right-hand side of the screen shows: **DNC**), press [**6**] (DNC ON/OFF) to deactivate it (the **DNC** letters disappear).

With options 0, 1, 2 and 3 it is possible to transfer machine parameters, the decoded M function table and the leadscrew error compensation table to a peripheral device.

The lower right-hand side of the CNC screen will show a directory of up to 7 part-programs of the CNC.

To do this, key in the desired number when the CNC requests the number of the program to be transferred and press [**ENTER**].

P00000 to P99990	Corresponding to part-programs
P99994 and P99996	Special user programs in ISO code
P99997	**For internal use and CANNOT be transmitted back and forth**
P99998	Used to associate texts to PLCI messages
P99999	Machine parameters and tables

Atention:

The part-programs cannot be edited at the peripheral device or computer.

The CRT will show the message: "**RECEIVING**" or "**SENDING**" during the program transfer and the message: "**PROGRAM NUM. P23256** (for example) **RECEIVED**" or "**SENT**" when the transmission is completed.

When the transmission is not correct, it will display the message: "Transmission error" and when the data received by the CNC is not recognized (different format) by the CNC, it will issue the message: "Incorrect data received".

The CNC memory must be unlocked in order to perform any data transmission; if not so, the CNC will return to the menu of the peripheral mode.

When transmitting from a peripheral device other than a FAGOR Floppy Disk Unit, the following aspects must be considered:

* The program must begin with a "NULL" character (ASCII 00) followed by "%" "program number" (for example %23256) and a "LINE FEED" character (LF).

* Blank spaces, the carriage-return key and the "+" sign are ignored.

* The program must end with either 20 "NULL" characters (ASCII 00) or with one "ESCAPE" character or with one "EOT" character.

* Press [**CL**] to cancel the transmission. The CNC will issue the message: PROCESS ABORTED".

FLOPPY DISK UNIT DIRECTORY

This option displays the programs stored on the disk inserted in the FAGOR Floppy Disk Unit and the number of characters (size) of each one of them.

It also shows the number of free characters available (free memory space) on the tape.

DELETE FLOPPY DISK UNIT PROGRAM

With this option it is possible to delete a program contained at the FAGOR Floppy Disk Unit.

The CNC requests the number of the program to be deleted. After keying in the desired number, press [**ENTER**].

Once the program has been deleted, the CNC will display the message: "PROGRAM NUM: P____ DELETED".

It also shows the number of free characters on the disk (free memory space).

3.8.2 DNC COMMUNICATIONS

To be able to use this feature, the DNC communication must be active (the upper right-hand side of the screen shows: DNC). To do this the corresponding parameters [P605(5,6,7,8); P606(8)] must be set accordingly and option [**6**] of the "**Peripherals**" mode selected if it was not active.

Once active and by using the **FAGORDNC** application software supplied, upon request, in floppy disks it is possible to perform the following operations from the computer:

. Obtain the CNC's part-program directory.
. Transfer part-programs and tables from and to the CNC.
. Delete part-programs at the CNC.
. Certain remote control of the machine.

Atention:

At the CNC any operating mode may be selected.

3.9 LOCK/UNLOCK

With this option it is possible to lock/unlock the machine parameters and the part-program memory.

To access this mode, press **[AUX]** and after selecting "Auxiliary Modes", press the key corresponding to the "LOCK/UNLOCK" option.

The codes used to do this are:

[P] 0000 [ENTER] Unlocks the machine parameters.

[P] 1111 [ENTER] Locks the machine parameters.

[BEG] 0000 [ENTER] Unlocks part-program memory.

[BEG] 1111 [ENTER] Locks part-program memory.

[P] F000 [ENTER] Erases the contents of all arithmetic parameters (data of the automatic operations) and sets them to "0".

3.10 EDITING PROGRAM 99996

Program 99996 is a special ISO-coded user program. It can be edited either in this operating mode or at a PC and then sent out to this CNC.

To select this option, press **[AUX]** and after selecting "Auxiliary Modes", press the key corresponding to "EXECUTION OF PROGRAM P99996".

The CNC displays the editing page for this program.

If the program is currently being edited, the CNC shows a group of program blocks (lines).

Use the [▲] [▼] to display the display the previous and following lines.

To edit a new line, follow this procedure:

1.- If the program line number appearing at the bottom of the screen is not the desired one, clear it by pressing **[CL]** and key in the desired line number.

2.- Key in all the pertinent data for that line and press **[ENTER]**.

The programming format to be used is described in the programming manual. The keys on the front panel may be used: **[X]**, **[Y]**, **[Z]**, **[S]**, **[F]**, **[N]** as well as: **[TOOL]** for T, [P] for P, [R] for R and [α] for A.

However, since some function keys are missing (G, M, I, K, etc.), an assisted editor is also available.

To access it, press **[AUX]**. After analyzing the syntax of what has been edited so far, the CNC will display, one by one, all the functions which can be edited at the time.

Press **[CL]** to delete characters.

To modify a previously edited line, proceed as follows:

1.- If the program line number appearing at the bottom of the screen is not the desired one, clear it by pressing **[CL]** and key in the desired line number.

2.- Press **[RECALL]**. The bottom of the screen of the CNC, editing area, will show the contents of that line.

3.- Use one of these methods to modify the contents:

a) Use the **[CL]** key to delete characters and edit it as described above.

b) Use the [◄] [►] keys to position the cursor over the section to be modified and use the **[CL]** key to delete characters or **[INC/ABS]** to insert data.

While in the data inserting mode, the characters behind the cursor appear blinking. It is not possible to use assisted programming (the **[AUX]** key).

Key in all the desired data and press **[INC/ABS]**. If the syntax of the new line is correct, the CNC will display it without blinking and, if not, it will show it blinking until it is edited correctly.

4.- Once the line has been modified, press **[ENTER]**. The CNC will assume it replacing the previous one.

To delete a program line, proceed as follows:

1.- If the program line number appearing at the bottom of the screen is not the desired one, clear it by pressing **[CL]** and key in the desired line number.

2.- Press **[DATA]** and the CNC will delete it from memory.

4. MACHINE PARAMETERS

Atention:

All unused machine parameters must be set to "0" to guarantee the proper functioning of this CNC.

It is recommended to save the machine parameters of the CNC at a peripheral device or computer in order to be able to recover them after their accidental loss.

Please note that some of the machine parameters mentioned here are described in greater detail in the chapter on "CONCEPTS" in this manual.

4.1 INTRODUCTION

On power-up, the CNC performs a system hardware test. When completed, it displays the model name and the message "GENERAL TEST PASSED" when successful and the corresponding error message if otherwise.

In order for the machine-tool to be able to properly execute the programmed instructions and recognize the interconnected elements, the CNC must "know" the specific data for the machine such as feedrates, acceleration ramps, feedback devices, etc.

This data is determined by the machine manufacturer and may be input via keyboard or via the RS232C serial line by setting the machine parameters.

To lock or unlock the access to the machine parameters, to the decoded M function table and to the leadscrew error compensation table, proceed as follows:

* Press **[AUX]** and after selecting the "Lock/Unlock" option of the of the "Auxiliary Modes" menu...

* Key in: "P1111" and press **[ENTER]** to lock the access or: "P0000" **[ENTER]** to unlock it.

When the access to the machine parameter table is locked, only those parameters related to the RS-232C serial communications line may be edited.

To enter the machine parameter values via the keyboard, press the following keystroke sequence:

[AUX]	(SPECIAL FUNCTIONS)
[5]	(AUXILIARY MODES)
[1]	(SPECIAL MODES)
[0] [1] [0] [1]	(Access code, password)
[1]	(MACHINE PARAMETERS)

4.2 OPERATING WITH PARAMETER TABLES

Once the machine parameter table has been selected, the operator may view the following or previous pages by means of the up and down arrow keys.

To view a particular parameter, key in the desired parameter number and press **[RECALL]**. The CNC will display the page corresponding to that parameter.

To EDIT a parameter, key in the desired number, press **[=]** and key in the value to be assigned to that parameter.

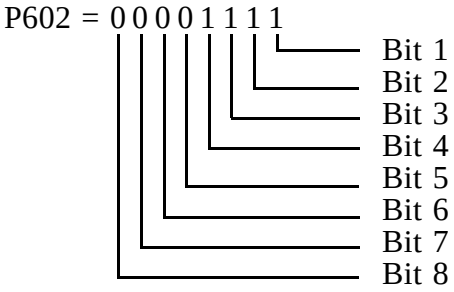
Depending on the type of machine parameter selected, it could be assigned one of the following types of values:

* A number	P111 = 30000
* A group of 8 bits	P602 = 00001111
* A character	P105 = Y

Once the value of the parameter has been keyed in, press **[ENTER]** so it is entered on the table.

If when pressing **[=]**, the parameter being edited disappears from the screen, it means that the machine parmeters are locked, therefore protected against modifications.

Every time a parameter bit is mentioned while describing the different machine parameters, refer to this nomenclature:



4.3 GENERAL MACHINE PARAMETERS

P5 AC frequency

Possible values: 50 Hz. and 60 Hz.

P99 Language

Determines the language used by the CNC to show texts and messages on the screen. 0 = Spanish. 1 = German. 2 = English. 3 = French. 4 = Italian.

P13 Measuring units (mm/inches)

It determines the measuring units assumed by the CNC for machine parameters, tool tables and work units at power-up, after executing M02 or M30 and after RESET.

0 = Millimeters (G71). 1 = Inches (G70).

P6 Theoretical or Real display

It determines whether the CNC will display the real axis position or the theoretical position.

0 (REAL)= The CNC displays the real position values (coordinates).
1 (THEO)= The CNC displays the theoretical position values (ignoring the following error).

P617(5), P605(6), P617(4) The X, Y, Z axis is a DRO axis.

It indicates whether the corresponding axis is treated as a normal axis (controlled by the CNC) or a DRO axis (moved externally).

0 = Normal axis. 1 = DRO axis.

P618(6), P618(5), P618(4) The X, Y, Z axis display

It indicates whether the corresponding axis is displayed on the CRT or not.

0 = It is displayed. 1 = It is **not** displayed.

P701 Number of tools

It is given by an integer between 0 and 98.

P626(1) The CNC displays the tool base position

It indicates whether the CNC displays the tool base or tool tip position when working with tool length compensation (G43).

0 = It displays the tool **tip** position.
1 = It displays the tool **base** position.

Atention:

When not working with tool length compensation (G44), the CNC always displays the tool base position.

P743 Subroutine to be executed before the T function
P745 Subroutine to be executed after the T function

The CNC takes these parameters into consideration when executing the ISO-coded user program 99996.

They indicate the number of the standard (non-parametric) subroutine to be executed by the CNC whenever a tool is selected in the execution of program 99996.

These standard subroutines must contain the tool selecting sequence.

They are set by an integer between 0 and 99. If set with a "0" value, the CNC assumes that no subroutine is to be executed.
Every time a new tool is selected in ISO code, the CNC behaves as follows:

1.-It executes the subroutine indicated by P743. If set to "0", the CNC will display the message: "TOOL CHANGE" (in English for all language versions) and it interrupts program execution.

2.-The CNC outputs the code for the new tool.

3.-It executes the subroutine indicated by P745. If set to "0", no subroutine is executed.

Atention:

When a subroutine is associated with the T function, nothing must be programmed in that block after the T function; otherwise, the CNC will issue the corresponding error.

The subroutines associated with the T function must be defined in one of the special ISO-coded user programs: P99994 or P99996.

P628(1) Display of the Following Error.

Enter: 0 = No, the Following Error is not displayed.
1 = Yes, the Following Error is displayed.

It is recommended to set to to "1" during machine set-up and then set it to "0" for normal operation.

P628(6) External device O1 may only be turned on/off when spindle stopped.

Enter: 0 = It may be turned on/off at any time, whether the spindle is on or off.

1 = It may be turned on/off only when the spindle is off.

4.3.1 INPUT/OUTPUT PARAMETERS

P605(8) Normal status of the Emergency output (pin 5 connector I/O 1)

It determines whether the emergency output is normally low or high.

0 = Normally low (0V). An emergency situation will set this output high (24V)

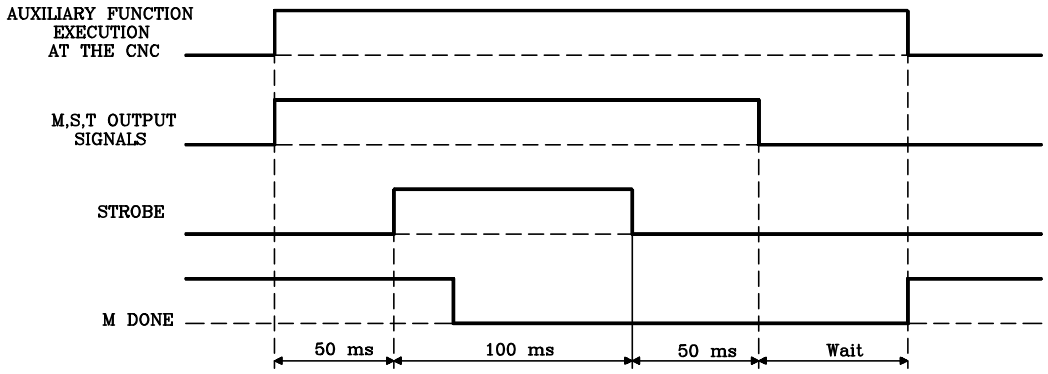
1 = Normally high (24V). An emergency situation will set this output low (0V)

P605(5) The CNC waits for the trailing edge of the M-DONE input signal

It indicates whether it is necessary or not to wait for the down flank (24V-to-0V transition) of the M-DONE signal (at pin 15 of connector I/O 1) in response to an "S STROBE", "T STROBE" or "M STROBE" so the CNC resumes the execution of such functions.

“P605(5)=0”

The CNC will send out to the electrical cabinet the BCD signals corresponding to the M, S or T code for a period of 200 milliseconds. Then, if the "M-DONE" signal is low (0V), it will wait for it to be set high (24V) in order to consider the M, S or T function done (completed).



“P605(5)=1”

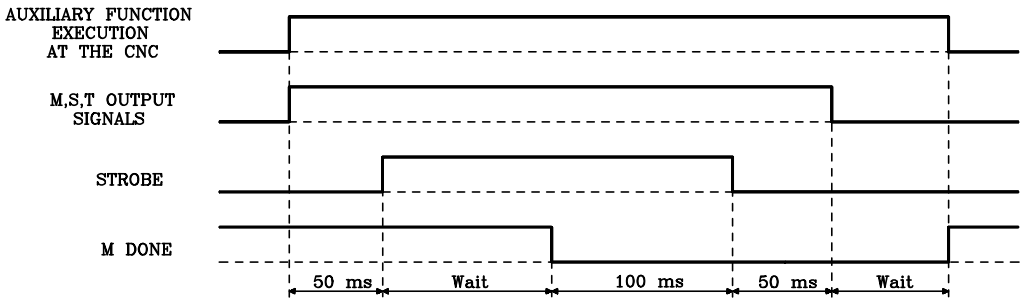
50 milliseconds after having sent the M, S or T BCD signals out to the electrical cabinet, it sends out the corresponding "Strobe" signal.

Then, if the "M-DONE" signal is high (24V), the CNC waits for it to be set low (0V).

Once the "M-done" signal is set low, the CNC continues maintaining the "Strobe" signal active for another 100 milliseconds.

After deactivating the Strobe signal, the M, S T BCD code signals are kept active for another 50 milliseconds.

After that time and if the "M-DONE signal is low, the CNC will wait until it becomes high so it can consider the auxiliary function M, S or T completed.



P609(7) Pin 17 of connector I/O 1 as RAPID TRAVERSE (fast feed)

It determines whether the signal input at pin 17 of connector I/O1 is treated as EXTERNAL CYCLE START or RAPID TRAVERSE.

0 = It is treated as EXTERNAL CYCLE START.
1 = It is treated as RAPID TRAVERSE.

If set as Rapid Traverse and while this input is active, the CNC will carry out all G01, G02 and G03 moves at 200% of the programmed feedrate F.

By the same token, in the JOG mode and while this input is kept active, the CNC will jog the axes in rapid G00.

P609(3) Pin 23 of connector I/O 2 as RESET output

Indicates whether there is or not a RESET output via pin 23 of connector I/O2.

0 = It is **not** used as RESET output.
1 = It **is** used as RESET output.

It must be borne in mind that this pin is also used as output 14 of the decoded M functions; therefore, it should not be set on the decoded M function table when this parameter is set to output a RESET signal (set to "1").

P611(1) Pin 24 of connector I/O 2 as program block in execution
P611(6) Pin 24 of connector I/O 2 as program P99996 in execution
P613(4) Pin 24 of connector I/O 2 as rapid move in progress

If machine parameter P611(1) is set to "1", the CNC sets this OUTPUT high (24V) every time a part-program block is being executed.

If machine parameter P611(6) is set to "1", the CNC sets this OUTPUT high (24V) whenever the special ISO-coded user program P99996 is being executed.

If machine parameter P613(4) is set to "1", the CNC sets this OUTPUT high (24V) whenever the CNC is executing a rapid positioning move

Care must be taken, when having one of these options, not to use the bit of the decoded M table corresponding to this output M13 since the CNC will activate it in all these cases.

0 = Used as output 13 of decoded M functions.
1 = Used as the corresponding indicator output and output 13 of the "M" functions.

When setting two or three of these parameters to "1", the CNC will only output one of them being P611(1) the one with the highest priority and P613(4) the one with the lowest.

P613(2) Pin 25 of connector I/O 2 as "Vertical axis movement" indicator output

It determines whether or not pin 25 of connector I/O2 is used to indicate the direction of the vertical axis movement. This output will be low (0V) for positive direction (count-up) or high (24V) for negative direction (count-down).

0 = It is **not** used as vertical axis moving direction indicator output.
1 = It **is** used as vertical axis moving direction indicator output.

It must be borne in mind that this pin is also used as output 12 of the decoded M functions; therefore, it should not be set on the decoded M function table when this parameter is set for the output to be used as vertical axis moving direction indicator (set to "1").

P617(8) M functions output in BCD or BINARY code

It determines whether the M function is output in BCD or Binary code via pins 20 thru 27 of connector I/O1.

0 = M function output in BCD code
1 = M function output in BINARY code

The significance or weight of each pin in both cases is as follows:

Pin	M in BCD WEIGHT	M in BINARY WEIGHT
27	1	1
26	2	2
25	4	4
24	8	8
23	10	16
22	20	32
21	40	64
20	80	128

For example: Depending on the type of code selected, the CNC will output the M41 as follows:

Pin	20	21	22	23	24	25	26	27
BCD	0	1	0	0	0	0	0	1
Binary	0	0	1	0	1	0	0	1

P609(5) Decoded M function code output in BCD or BINARY.

When executing an M function which is decoded on the table, the CNC activates the decoded outputs of connector I/O2.

This parameter determines whether or not the CNC activates the M-BCD outputs of connector I/O1 (pins 20 thru 27) **besides** the decoded M outputs of connector I/O2.

0 = It **also** outputs the M function in BCD or BINARY code.
1 = It **does not** output the M function in BCD or BINARY code.

P602(8), P602(7), P602(6), P602(5), P603(1) Feedback alarm cancellation of the A1, A2, A3, A4 and A5 connectors

The CNC will show the axis feedback alarm when not receiving all its corresponding feedback signals or when any of them is not within the permitted levels.

This parameter indicates whether this feedback alarm is to be cancelled or not.

0 = The feedback alarm for the corresponding axis is **not** cancelled.
1 = The feedback alarm for the corresponding axis **is** cancelled.

If the feedback system being used only utilizes 3 square-wave signals (A, B and Io), the corresponding parameter must be set to "1" (feedback alarm for that axis canceled).

4.3.2 MACHINE PARAMETERS FOR THE HANDWHEELS

P612(1) There is an electronic handwheel associated with the X axis.
P626(8) There is an electronic handwheel associated with the Y axis.
P627(8) There is an electronic handwheel associated with the Z axis.

0 = There is **no** electronic handwheel for this axis.
1 = There **is** an electronic handwheel for this axis.

P613(1) The connected handwheel is a FAGOR 100P model

This parameter is relevant when using a single handwheel associated to the X axis and it indicates whether or not it is a FAGOR 100P model with axis selector button.

0 = It is **not** a FAGOR 100P model.
1 = It **is** a FAGOR 100P model.

P612(2) Counting direction of the X axis handwheel.
P626(2) Counting direction of the Y axis handwheel.
P627(2) Counting direction of the Z axis handwheel.

Define the counting direction of the electronic handwheel. If correct, leave it as is; if not, change it.

Possible values: "0" and "1"

P612(3) Feedback units for the X axis handwheel.
P626(3) Feedback units for the Y axis handwheel.
P627(3) Feedback units for the Z axis handwheel.

Indicate whether the feedback pulses provided by the handwheel correspond to millimeters or inches.

0 = millimeters.
1 = inches.

P612(4,5) Feedback resolution of the X axis handwheel.
P626(4,5) Feedback resolution of the Y axis handwheel.
P627(4,5) Feedback resolution of the Z axis handwheel.

Indicate the counting resolution of the electronic handwheel. That is, the distance corresponding to each square feedback pulse. The possible values are:

bit (4)	bit (5)	Resolution
0	0	0.001 mm or 0.0001 inch
0	1	0.002 mm or 0.0002 inch
1	0	0.005 mm or 0.0005 inch
1	1	0.010 mm or 0.0010 inch

P612(6) Multiplying factor for the X axis handwheel signals.
P626(6) Multiplying factor for the Y axis handwheel signals.
P627(6) Multiplying factor for the Z axis handwheel signals.

Indicate the x2 or x4 multiplying factor that the CNC will apply to the feedback signals provided by the electronic handwheel.

0 = x4.
1 = x2.

Example: If the X axis handwheel has been set as follows:

P612(3) = 0	Millimeters
P612(4) = 0 & P612(5) = 0	Resolution 0.001 mm.
P612(6) = 0	x4

The F.O.S. is at "x100" position.

The X axis will move 0.001mm x4 x100 = 0.4 mm per handwheel pulse received.

P628(5) The handwheels operate at all "handwheel" positions of the switch

0 = The electronic handwheels **only** operate at the positions of the Feedrate Override Switch indicated by the symbol.

1 = The electronic handwheels operate **at all** the **positions** of the Feedrate Override Switch. At those positions indicated by the symbol, the CNC applies the selected pulse multiplying factor (x1, x10, x100) and applies a x1 factor at the other positions of the switch.

P628(8) The machine uses mechanical handwheels

0 = The machine **does not use** mechanical handwheels.

1 = The machine **uses** mechanical handwheels.

When using mechanical handwheels, "P628(8) =1", machine parameters P105, P205 and P305 must be set to "**N**" so the axes are not continuously controlled. In other words, they **are not held in position** once they reach their in-position (dead-band) zone.

P815 Delay before opening the loop

This parameter only applies when the machine uses mechanical handwheels and, therefore, machine parameters "P628(8)=1", "P105=N", "P205=N", and "P305=N".

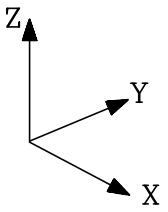
It indicates the time delay from the moment the axes reach position to when the CNC opens their position loops.

It is expressed by an integer between 0 and 255.

Value of "0" = No delay is applied.
Value of "1" = 10 milliseconds.
Value of "10" = 100 milliseconds.
Value of "255" = 2550 milliseconds.

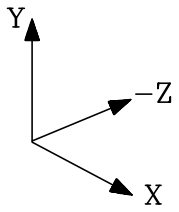
4.3.3 PARAMETERS RELATED TO THE OPERATING MODE

P609(8) Graphic representation of coordinate system



P609(8)=0

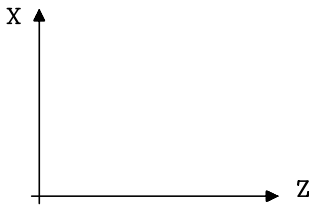
Mill model



P609(8)=1

Boring Mill model

P605(4) XZ plane representation



P605(4)=0

P605(4)=1

P628(7) Interpolations with the Z axis

0 = Any two- or three-axis interpolation is possible (XY, XZ, YZ and XYZ)

1 = **No** interpolation involving the **Z axis** is possible. When programming a movement which includes the Z axis, the CNC will first move the Z axis and then the XY axes.

P618(1) Disabling the CYCLE START key

It indicates whether the CYCLE START key of the operator panel is cancelled (ignored by the CNC) or not.

0 = The CYCLE START key is **not disabled.**
1 = The CYCLE START key **is disabled** (ignored by the CNC).

P606(3) M30 when switching to JOG mode

It indicates whether the CNC must generate an M30 automatically or not when switching to the JOG mode.

0 = M30 is **not** generated.
1 = M30 **is** generated.

P12 Continuous or pulsating axis jog

It indicates whether the axes are jogged **while** their corresponding jog keys are pressed (pulsating) or their movements are maintained until the CYCLE STOP key or another jog key is pressed (continuous).

0 = Continuous mode. The axis starts moving when its corresponding jog key is pressed and it stops when the CYCLE STOP key or another jog key is pressed. In this latter case, the CNC will move the new selected axis in the chosen direction until the CYCLE STOP key or another jog key is pressed.

1 = Pulsating mode. The axis is jogged **as long as** its corresponding jog key is maintained pressed.

P609(6) Maximum incremental JOG move

It indicates the maximum distance the axes can be jogged when selecting one of the JOG positions of the Feedrate Override Switch on the operator panel (positions 1, 10, 100, 1000, 10000).

0 = Limited to 10 mm. or 1 inch.
1 = Limited to 1 mm. or 0.1 inch.

P606(2) Maximum value of the Manual Feedrate Override

It indicates the maximum feedrate override value to be selected by the Manual Feedrate Override Switch at the operator panel.

0 = Possible up to 120%.
1 = Limited to 100% even when selecting the 110% and 120% switch positions.

P4 The Manual Feedrate Override switch active in G00

It indicates whether it is possible or not to override the axis feedrate by this switch when moving in G00 (rapid positioning).

NO The feedrate override switch is ignored when in G00.

YES The feedrate override switch is active (not ignored) when in G00 applying a range from 0% to **100%** of the maximum feedrate set by machine parameters P111, P211, P311, P411 and P511 even at 110% and 120% positions.

P613(5) G05 or G07 active on power-up

It indicates whether the CNC assumes function G05 (round corner) or G07 (square corner) on power-up, after M02, M30, EMERGENCY or RESET.

0 = G07 (square corner).
1 = G05 (round corner).

P715 Dwell between blocks in G07 (square corner)

It defines the dwell applied to motion blocks in G07.

It is given by an integer between 0 and 255.

Value 0 = No dwell.
Value 1 = 10 msec.
Value 10 = 100 msec.
Value 255 = 2550 msec.

P611(5) Feedrate units in G94

It determines the F programming units when function G94 is active.

0 = 1 mm./minute or 0.1 inch/minute.
1 = 0.1 mm./minute or 0.01 inches/minute.

If parameter "P611(5)=1", it is working in mm and F0.1 is programmed, the applied feedrate will be F0.01 mm/min.

It must be borne in mind that the machine parameters corresponding to the maximum programmable feedrate **F0** (P110/210/310), the maximum feedrate in **G00** (P111/211/311), the **home searching** feedrate (P112/212/312) and the **unidirectional approach** feedrate (P801) **are not affected by this parameter**. They are expressed in 1 mm/min or 0.1 inch/min units.

P610(1) FEED-HOLD in G47

It indicates whether the CNC stops the movement of the axes while the FEED-HOLD input is active while in G47 (single block treatment).

0 = The FEED-HOLD input **does not** stop the axes.
1 = The FEED-HOLD input stops the axes.

P628(4) The "rapid jog" key applies a feedrate override range over 100 %

This parameter indicates the type of jogging feedrate override that will be applied while pressing key

0 = While keeping this key pressed, the CNC will apply a feedrate override amount according to the table below.

% selected	0	2	4	10	20	30	40	50	60	70	80	90	100	110	120
% applied	0	102	104	110	120	130	140	150	160	170	180	190	200	200	200

When this key is released, the amount of override will return to its face value (0 through 120%).

1 = While keeping this key pressed, the CNC will apply the maximum feedrates set by machine parameters: P111, P211 and P311.

It may be interesting to set this parameter P628(4) to "0" on short travel machines and to "1" on large ones.

4.3.4 MACHINE PARAMETERS FOR THE RS232C SERIAL LINE

P0 Transmission speed (baudrate)

It determines the transmission baudrate used in communications between the CNC and the peripheral devices.

It is given by an integer (9600 maximum) and in baud units.

Typical values:

110, 150, 300, 600, 1200, 2400, 4800, 9600

P1 Data bits per transmitted character

It determines the number of data bits used in each transmitted character.

Possible values:

7 = Only the 7 least significant bits (out of 8) are used. Assign this value when transmitting standard ASCII characters.

8 = All 8 bits of the transmitted character are used. Assign this value when transmitting special characters (ASCII code over 127).

P2 Parity

It determines the type of parity check used in the transmission.

Possible values:

0 = None.
1 = ODD parity.
2 = EVEN parity.

P3 Stop bits

It determines the number of stop bits used at the end of the transmitted word.

Possible values:

1 = 1 stop bit.
2 = 2 stop bits.

P607(3) DNC

It indicates whether the CNC can work with the DNC protocol or not.

0 = DNC function **not** available.
1 = DNC function **available**.

P607(4) Communication settings for the FAGOR Floppy Disk Unit or Cassette.

P607(4)=1 For Floppy Disk Unit. The CNC uses the values set for machine parameters P0, P1, P2 and P3.

P607(4)=0 For Cassette Unit. The CNC does not modify the P0, P1, P2 and P3 settings but uses the values corresponding to the FAGOR Cassette Unit.

Baudrate: 13,714 Baud
Data bits: 7
Parity: Even
Stop bits: 1

Atention:

In DNC and Peripheral communications, use the settings of machine parameters P0, P1, P2 and P3.

P607(5) DNC protocol active on power-up

It indicates whether the DNC protocol is active on CNC power-up or not.

0 = DNC **not** active on power-up.
1 = DNC **active** on power-up.

P607(6) The CNC does not abort DNC communication (program debugging)

The CNC offers a safety system that aborts DNC communications whenever:

* More than 30 seconds elapse without receiving a character while in the reception mode.

* More than 3 incorrect acknowledgments or non-acknowledgments occur in a row while in transmission mode.

This parameter can be used in order to be able to debug a user communications program without the CNC aborting the communication.

0 = The CNC aborts communications.
1 = The CNC does **not** abort communications (Debug mode).

P607(7) Status report by interruption

It indicates whether the "status report by interruption" is active or not while in DNC mode.

0 = It is **not** active.
1 = It is active.

A more detailed explanation on this function can be found in the "DNC COMMUNICATIONS PROTOCOL FOR THE 8025 CNC" manual.

5. *MACHINE PARAMETERS FOR THE AXES*

Atention:

Observe that some of the parameters mentioned in this chapter are also described in more detail in the chapters on "power and machine interface" and "concepts" in this manual.

P100, P200, P300 Sign of the analog voltage for X, Y, Z axes

It determines the sign of the analog voltage for the axis servo drive. If correct, leave it as is and change it if otherwise.

Possible values: “0” and “1”.

P101, P201, P301 Sign of the X, Y, Z axis feedback

It determines the counting direction of the axis. If correct, leave it as is and change it if otherwise.

Possible values: “0” and “1”.

Observe that when changing this parameter, the one corresponding to the sign of the analog voltage must also be changed to prevent the axis from running away (P100, P200, P300).

P102, P202, P302 X, Y, Z axis jogging direction

It establishes the axis jogging direction by means of the JOG keys of the operator panel.

It determines the counting direction of the axis. If correct, leave it as is and change it if otherwise.

Possible values: “0” and “1”.

5.1 PARAMETERS RELATED TO AXIS RESOLUTION

The section on "Axis resolution" in the chapter on "concepts" of this manual describes how these parameters may be used.

P103, P203, P303 X, Y, Z axis feedback resolution

They indicate the counting resolution for the axis.
Possible values for square-wave signals:

1 = Resolution of 0.001 mm, 0.0001 inch.
2 = Resolution of 0.002 mm, 0.0002 inch.
5 = Resolution of 0.005 mm, 0.0005 inch.
10 = Resolution of 0.010 mm, 0.0010 inch.

The units used depend on the setting of the following parameters:

P604(4,3,2) In millimeters or inches.

P622(1), P622(2), P622(3) Counting resolution for X, Y, Z, axis with sine-wave feedback

When using sine-wave feedback signals, the CNC considers these parameters as well as P103, P203 and P303, to set the axis resolution.

Possible values for P103, P203, and P303 with P622(1), P622(2), P622(3) = 0:

5 = Resolution of 0.001 mm, 0.0001 inch.
10 = Resolution of 0.002 mm, 0.0002 inch.

Possible values for P103, P203 and P303 with P622(1), P622(2), P622(3) = 1:

1 = Resolution of 0.001 mm, 0.0001 inch.
2 = Resolution of 0.002 mm, 0.0002 inch.
5 = Resolution of 0.005 mm, 0.0005 inch.
10 = Resolution of 0.010 mm, 0.0010 inch.

P604(4), P604(3), P604(2) X, Y, Z feedback units

They indicate the units of the feedback pulses for the corresponding axis.

0 = Millimeters.
1 = Inches.

P106, P206, P306 X, Y, Z axis feedback signal type

They indicate the type of feedback signals being used.

0 = Square-wave feedback signals
1 = Sine-wave feedback signals

The CNC **always** applies a x5 multiplying factor to the sine-wave feedback signals.

P604(8), P604(7), P604(6) Multiplying factor for X, Y, Z axis feedback signals

It indicates whether the CNC applies a x2 or x4 multiplying factor to the feedback signals of the axes.

0 = It applies a x4 factor.
1 = it applies a x2 factor.

When using FAGOR linear transducers (scales), set the corresponding parameters to "0".

Setting examples for the X axis:

Using square-wave linear transducers (scales):

Since the CNC applies either a x2 or x4 multiplying factor, a linear transducer must be selected which has a signal period of twice or four times the desired resolution.

Desired resolution		P103	P604(8)	Feedback Signal period	FAGOR Linear transducer
P604(4)=0	P604(4)=1				
0.001 mm	0.0001 inch	1	x2 P604(8)=1	0.002 mm	
			x4 P604(8)=0	0.004 mm	CX, CVX, MX
0.002 mm	0.0002 inch	2	x2 P604(8)=1	0.004 mm	CX, CVX, MX
			x4 P604(8)=0	0.008 mm	
0.005 mm	0.0005 inch	5	x2 P604(8)=1	0.010 mm	
			x4 P604(8)=0	0.020 mm	CT, CVT, MT, MVT, FT
0.010 mm	0.0010 inch	10	x2 P604(8)=1	0.020 mm	CT, CVT, MT, MVT, FT
			x4 P604(8)=0	0.040 mm	

Using sine-wave linear transducers and P622(1)=1:

Besides the x2 or x4 selected by P604(8), the CNC applies an additional x5 factor to the sinewave signals. Therefore, a transducer must be chosen which has a feedback signal period 10 or 20 times the desired resolution.

If parameter P622(1)=1, it is possible to obtain resolution of 1, 2, 5 and 10 microns or ten-thousandths of an inch.

Desired resolution		P103	P604(8)	Feedback signal period	FAGOR Linear transducer
P604(4)=0	P604(4)=1				
0.001 mm	0.0001 inch	1	x2 P604(8)=1	0.010 mm	
			x4 P604(8)=0	0.020 mm	CC,CVC,CVS,MC,MVC,
0.002 mm	0.0002 inch	2	x2 P604(8)=1	0.020 mm	CC,CVC,CVS,MC,MVC,
			x4 P604(8)=0	0.040 mm	
0.005 mm	0.0005 inch	5	x2 P604(8)=1	0.050 mm	
			x4 P604(8)=0	0.100 mm	FS
0.010 mm	0.0010 inch	10	x2 P604(8)=1	0.100 mm	FS
			x4 P604(8)=0	0.200 mm	

Using sine-wave linear transducers and P622(1)=0:

Besides the x2 or x4 selected by P604(8), the CNC applies an additional x5 factor to the sinewave signals. Therefore, a transducer must be chosen which has a feedback signal period 10 or 20 times the desired resolution.

If parameter P622(1)=0, it is possible to obtain resolution of 1 and 2 microns or ten-thousandths of an inch.

Desired resolution		**P103**	**P604(8)**	**Feedback signal period**	**FAGOR Linear transducer**
P604(4)=0	P604(4)=1				
0.001 mm	0.0001 inch	5	x2 P604(8)=1	0.010 mm	
			x4 P604(8)=0	0.020 mm	**CC,CVC,CVS,MC,MVC,**
0.002 mm	0.0002 inch	10	x2 P604(8)=1	0.020 mm	**CC,CVC,CVS,MC,MVC,**
			x4 P604(8)=0	0.040 mm	

P603(8), P603(7), P603(6) Binary encoder on X, Y, Z axes

It indicates whether the corresponding axis has a BINARY encoder (1024/2048 lines per turn) or not.

0 = It **is not** a binary encoder.
1 = It is a binary encoder.

P610(8), P610(7), P610(6) Equivalence of the binary encoder for X, Y, Z axes

This parameter is to be set when using a binary encoder (1024 or 2048 pulses) in place of one with 1000 or 1250 lines to obtain the desired resolution.

By setting this parameter, the CNC will adapt the encoder pulse-count as follows:

0 = It will treat the 1024-count binary encoder as a 1250-count and the 2048-count binary encoder as a 2500-count encoder.
1 = It will treat the 1024-count binary encoder as a 1000-count and the 2048-count binary encoder as a 2000-count encoder.

To calculate the axis resolution (P103, P203, P303) use the equivalent number of pulses selected here (1000, 1250, 2000, 2500).

The usefulness of the binary encoders is obvious since the same encoder can be utilized on two different types of leadscrews (for example 4-pitch and 5-pitch) without having to stock two different encoder models.

$$\text{Encoder} = \frac{\text{Leadscrew pitch}}{\text{Multiplying factor x Resolution}} = \frac{\text{1/4 inch/turn}}{\text{x4 x 0.0001 inch/pulse}} = 1250 \text{ pulses/turn}$$

$$\text{Encoder} = \frac{\text{Leadscrew pitch}}{\text{Multiplying factor x Resolution}} = \frac{\text{1/5 inch/turn}}{\text{x2 x 0.0001 inch/pulse}} = 1000 \text{ pulses/turn}$$

5.2 PARAMETERS RELATED TO THE ANALOG OUTPUTS

The section on "Adjustment of the axes" in the chapter on "concepts" of this manual describes how these parameters may be used.

P117, P217, P317 Minimum analog for X, Y, Z axes

They determine the minimum analog values for the axes.

It is given by an integer between 1 and 255.

Value 1 = 2.5 mV.
Value 10 = 25.0 mV. (10 x 2.5)
Value 255 = 637.5 mV. (255 x 2.5)

P104, P204, P304 Enable-Analog delay for X, Y, Z axes

They indicate whether there is 400 msec delay from the time the axis ENABLE signal is activated to when its analog voltage is output.

0 = There is **no** delay.
1 = There is delay.

This parameter is used when the axis is not continuously controlled (held in position by the CNC) and some time it is required to deactivate certain devices such as a holding brake, etc.

P118, P218, P318 In-position zone (dead-band) for X, Y, Z

They define the in-position zone (to either side of the programmed coordinate value) where the CNC considers the axis to be in position.

It is always expressed in microns regardless of the work units being used.

Possible values: 0 thru 255 microns.

P105, P205, P305 Continuous control of the X, Y, Z axes

They determine whether or not the axis is held in position by the CNC by keeping its ENABLE signal ON when reaching its target position.

1 = It is continuously controlled (ENABLE on when in position).
0 = It is **not** continuously controlled (ENABLE off when in position)

5.3 PARAMETERS RELATED TO TRAVEL LIMITS

The section on "Adjustment of the axes" in the chapter on "concepts" of this manual describes how these parameters may be used.

P107, P207, P307 Positive travel limit for X, Y, Z axes
P108, P208, P308 Negative travel limit for X, Y, Z axes

They establish the positive and negative travel limits for the axes. These distances are referred to machine reference zero (home).

Possible values: ± 8388.607 millimeters.
± 330.2599 inches.

If both limits are assigned the same value (for example: 0), the CNC will not allow to move the axis.

In JOG mode, and for safety reasons, it is possible to move the axis only up to 100 microns from the travel limits set by these parameters.

5.4 FEEDRATE RELATED PARAMETERS

P110, P210, P310 Maximum programmable feedrate for X, Y, Z

They determine the maximum programmable feedrate F0 for the axes.

Possible values: 1 thru 65,535 mm./minute.
1 thru 25,800 0.1inch/minute.

P111, P211, P311 G00 feedrate for X, Y, Z

They determine the rapid positioning feedrate (G00).

Possible values: 1 thru 65,535 mm./minute.
1 thru 25,800 0.1inch/minute.

P729 Maximum feedrate F for circular interpolations

It determines the maximum feedrate for circular interpolations. This value is a function of the arc radius according to the formula:

$$\text{F maximum} = \frac{\text{P729 x Radius}}{0.085}$$

It is given by an integer between 0 and 255. If set to "0", there will be no feedrate limitation.

Example:

We set parameter P729 = 17 so the feedrate on a 15mm-radius arc is limited to 3000 mm/min.

If we, then, program a 100mm-radius arc, the maximum feedrate for that arc will be:

$$\text{F maximum} = \frac{\text{P729 x Radius}}{0.085} = \frac{17 \times 100}{0.085} = 20000 \text{ mm/min}$$

P708 Feedrate override when the analog voltage reaches 10V.

It indicates the Feedrate override (%) that the CNC applies when the analog voltage of an axis reaches 10V.

It is given by an integer between 0 and 128.

Value 0 = No % is applied.
Value 32 = 25 %
Value 64 = 50 %
Value 128 = 100 %

This parameter makes the CNC "wait" for the axis to catch up, reducing its analog voltage; thus preventing from issuing the corresponding following error alarms.

P714 Error if axis feedrate is not between 50% & 200% of programmed F

It indicates whether or not the CNC verifies that the actual axis feedrate is between 50% and 200% of the programmed F value.

This parameter is assigned a time period in which the axis feedrate is allowed to be out of this 50%-200% range.

It is expressed by an integer between 0 and 255.

Value	0 =	The actual feedrate is not monitored in this sense.
Value	1 =	10 msec.
Value	10 =	100 msec.
Value	255 =	2550 msec.

P615(6) Feedrate in inches/minute

This parameter is used when working in inches (G70).

It indicates whether the axis feedrate units are inches/min. or 0.1 inch/min.

0 = Feedrate value in 0.1inch/min. (example: F10 = 1 inch/min.)
1 = Feedrate value in inches/min. (example: F10 = 10 inches/min.)

5.5 PARAMETERS RELATED TO AXIS CONTROL

The section on "Gain adjustment " in the chapter on "concepts" of this manual describes how these parameters may be used.

P114, P214, P314 Proportional gain K1 for X, Y, Z axes

They set the analog output for 1 micron of following error.

It is given by an integer between 0 and 255 in such a way that a value of 64 corresponds to an analog voltage of 2.5mV.

$$\text{Analog (mV)} = K1 \times \text{Following error (microns)} \times \frac{2{,}5\text{mV.}}{64}$$

P115, P215, P315 Gain break-point for X, Y, Z axes

They define the following error value from where the proportional gain K2 takes over and K1 is no longer applied.

It is recommended to set these parameters to a value slightly greater than the following error corresponding to the maximum machining feedrate F0.

Value range: 1 thru 32766 microns
1 thru 12900 ten-thousandths of an inch (= 1.29 inches)

P116, P216, P316 Proportional gain K2 for X, Y, Z axes

They set the analog output for 1 micron of following error from the gain break-point on.

It is given by an integer between 0 and 255 in such a way that a value of 64 corresponds to an analog voltage of 2.5mV.

Analog = (K1 x Ep) + [K2 x (Following error -Ep)]

Where Ep is the value of the gain break-point.

It is recommended to set these parameters to a value between 50% and 70% of K1 in order to prevent jerky transitions between K1 and K2 or between machining feedrates and rapid positioning (G00).

Refer to the sections on "adjustment of the proportional gain" in the chapter about "MACHINE AND POWER INTERFACE".

P611(8) G00 and F00 always with proportional gain K2 from a set gain break-point of 256 microns

It determines whether the gain break-point being applied is the one set by the user or set at 256 microns.

0 = Gain break-point set by the user
1 = Gain break-point set at 256 microns.

P726 Recovery of programmed position of axes without continuous control.

It determines how the CNC behaves regarding the non-continuously controlled axes once they reach the programmed position.

It is assigned an integer value between 0 and 255.

Once the programmed position is reached, the axis is "free" since its enable signal disappears and it is no longer controlled by the CNC. However, depending on the value given to this parameter, it behaves as follows:

P726 = 0

If the axis drifts out of position a distance greater than 16 times the in-position value (P118, P218, P318), the CNC will issue the corresponding following error message.

P726 = other than zero.

If the axis drifts out of position a distance greater than “P726”/2 times the in-position value (P118, P218, P318), the CNC activates the corresponding enable signal in order to recover the drifted distance.

5.6 PARAMETERS RELATED TO MACHINE REFERENCE ZERO

The section on "Reference systems" in the chapter on "concepts" of this manual describes how these parameters may be used.

P119, P219, P319 Coordinate of Machine Reference Zero for X, Y, Z axes

They determine the distance from this point to the Machine Reference Zero.

Possible values: ± 8388.607 millimeters.
± 330.2599 inches.

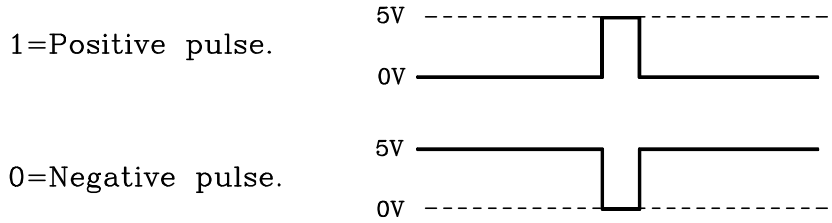
P623(8), P623(7), P623(6) Home searching direction for X, Y, Z axes

They determine the direction of the axis move while searching the Machine Reference Point.

0 = Positive direction.
1 = Negative direction.

P600(8), P600(7), P600(6) Type of machine reference pulse for X, Y, Z axes

They define the type of reference pulse (marker, Io) of the feedback device being used.



The marker pulses of FAGOR linear transducers are negative (parameter=0) and FAGOR rotary encoders have a positive marker pulse (parameter=1) per revolution.

P602(4), P602(3), P602(2) Home switch for X, Y, Z axes

They indicate whether or not a home switch is being used for the axis home search.

0 = Yes, the axis has a home switch.
1 = No, the axis does not have a home switch.

P112, P212, P312 1st home searching feedrate for X, Y, Z axes
P810, P811, P812 2nd home searching feedrate for X, Y, Z axes

They determine the feedrates used during home search.

The axis will move at the 1st feedrate until the home switch is pressed and, then, at the 2nd feedrate until the marker pulse is found on the feedback device.

Possible values: 1 thru 65,535 mm./minute
1 thru 25,800 0.1inch/minute.

When setting the 2nd feedrate to "0", the axis will move at 100 mm/min. (about 4 inches/min).

P611(2) Home search required on power-up

It determines whether it is required or not to perform the home search on all the axes after powering the CNC up.

0 = It is **not** required.
1 = It **is** required.

Being this parameter set to "1", the CNC will issue the corresponding error message when attempting to execute a part-program in AUTOMATIC, SINGLE BLOCK or TEACH-IN mode.

P804 Jogging feedrate prior to referencing (homing) the axes of the machine

This parameter is used when "P611(2)=1", thus being mandatory to home all the axes after machine power-up.

It indicates the feedrate being applied when moving the axes with the Jog keys and the handwheels if the axes have not been referenced (homed) yet.

Possible values: 1 thru 65,535 mm./minute
1 thru 25,800 0.1inch/minute.

P606(4) The home search generates an M30

It indicates whether the CNC generates an M30 automatically when executing the axes homing function.

0 = M30 is **not** generated.
1 = M30 **is** generated.

5.7 PARAMETERS FOR ACCELERATION/DECELERATION OF THE AXES

The section on "Gain adjustment " in the chapter on "concepts" of this manual describes how these parameters may be used.

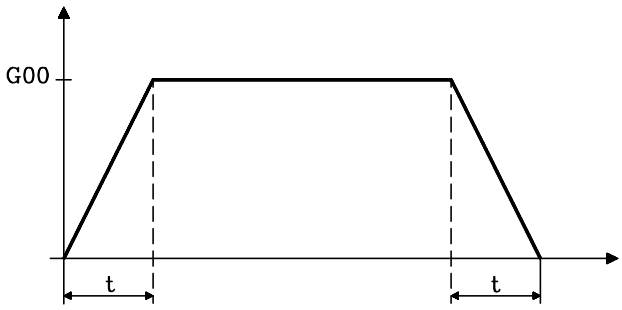
5.7.1 LINEAR ACCELERATION/DECELERATION

This type of acc./dec. is applied mainly on G00 and F00 moves although it is also possible to use it in G01 moves.

P721, P722, P723 ACC./DEC. Control of the X,Y, Z axes.

In order to avoid abrupt start-ups and brakes of the machine, it is possible to define some acceleration and deceleration ramps.

These parameters define the time that each axis needs to reach the positioning feedrate (machine parameters P111, P211, P311) while accelerating. This acceleration time will be the same as the deceleration time.



It is given by an integer between 0 and 255.

Value of 0 = There is no Acceleration/deceleration control.
Value of 1 = 0.020 seconds.
Value of 10 = 0.200 seconds. (10 x 0.02)
Value of 255 = 5.100 seconds. (255 x 0.02)

During a linear interpolation G01 at F0, the CNC applies the longest of the acc./dec. times assigned to the axes involved in the move.

Note: No acc./dec. will be applied on circular interpolations (G02, G03).

P613(7) ACC./DEC. in all linear interpolations (G01).

It indicates whether the acc./dec. ramps (P721, P722, P723) are to be applied on all linear interpolations or only when they are carried out at the maximum feedrate set by machine parameters P110, P210, P310.

0 = Acc./Dec. applied only in linear interpolations at maximum feedrate.
1 = Acc./Dec. applied in **all** linear interpolation (at any feedrate).

P620(2) Acceleration/deceleration in G05 (corner rounding)

It indicates whether or not the acc./dec. ramps are applied in blocks with G05 active (corner rounding).

0 = Yes. Acc./dec. is being applied.
1 = No. Acc./dec. is not being applied.

5.7.2 BELL-SHAPED ACCELERATION/DECELERATION

This type of acceleration may be applied on all kinds of movements, G00, G01, G02, etc and for any type of feedrate F.

P624(8) Bell-shaped acceleration/deceleration

It is to be used on high-speed machines.

The acc./dec. ramps are applied onto all kinds of movement: rapid, linear interpolation, **arcs**, etc.

0 = No. This type of acc./dec. is not applied.
1 = Yes. This type of acc./dec. is applied.

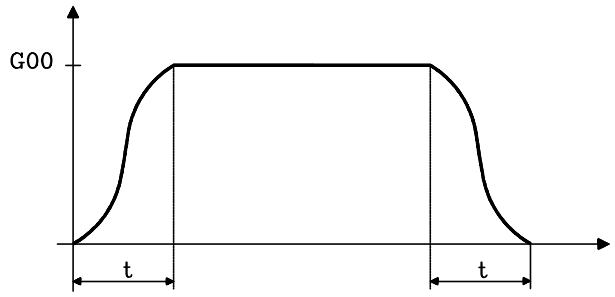
This parameter cancels the other acc./dec. parameters: P721, P722, P723, P724, P728 and P613(7).

Note that the acc./dec. ramps set by this parameter and P744 is common to all the axes.

P744 Duration of the Bell-shaped Acc./Dec. ramp

This parameter will be used when machine parameter “P624(8)=1”.

It defines the time needed by the axis to reach the selected feedrate (when accelerating). This time period is the same for the deceleration stage and common to all the axes of the machine.



It is given by an integer between 0 and 255.

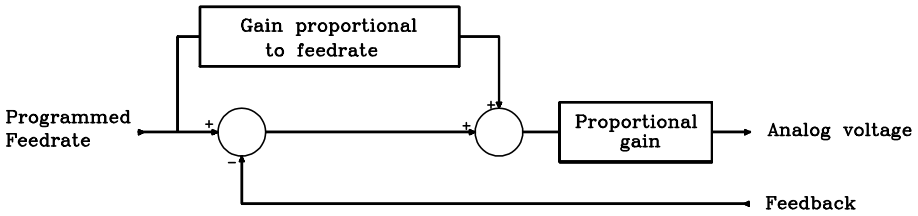
Value of 0 = There is no Acc./Dec. common to all the axes
Value of 1 = 0.010 seconds.
Value of 10 = 0.100 seconds. (10 x 0.01)
Value of 255 = 2.550 seconds. (255 x 0.01)

5.7.3 FEED-FORWARD GAIN

P732, P733, P734 FEED-FORWARD gain for X, Y, Z axes.

With the Feed-forward gain, which is proportional to the feedrate, it is possible to improve the positioning loop minimizing the amount of following error. **However it is not recommended when acceleration/deceleration ramps are not being used.**

These parameters define the % of analog voltage due to the programmed feedrate. It is given by an integer between 0 and 255.



The value which will be added to the following error is (Kf x F/6) where F is the programmed feedrate and Kf is:

* The value of this parameter in the case of linear acc/dec.
  For example, for the X axis: "Kf = P732"

* An eighth of the value assigned to this parameter in the case of bell-shaped acceleration/deceleration.
  For example, for the X axis: "Kf = P732/8"

The CNC will apply the proportional gain (K1 and K2) to the value resulting from the addition of the following error plus the value selected by means of the feed-Forward gain.

When the value resulting from the addition is smaller than the value assigned to the gain break-point, the CNC will apply the following formula:

Analog = K1 x [Following error + (Kf x F/6)]

And when the value resulting from the addition is greater than the value of the gain break-point:

Analog = (K1 x Ep) + {K2 x [Following error + (Kf x F/6) - Ep]}

Where “Ep” is the gain break-point value assigned to the corresponding parameter.

5.8 LEADSCREW RELATED PARAMETERS

With this CNC it is possible to compensate for leadscrew error as well as for its backlash when reversing movement direction.

5.8.1 LEADSCREW BACKLASH

P109, P209, P309 Backlash compensation for X, Y, Z axes

They indicate the amount of leadscrew backlash. When using linear transducers (scales), the corresponding parameter must be set to "0".

It is given in microns regardless of the work units being used.

Possible values: 0 thru 255 microns.

P624(1), P624(2), P624(3) Sign of the backlash for X, Y, Z axes

They define the sign of the backlash compensation value indicated by parameters P109, P209, P309.

0 = Positive.
1 = Negative.

P113, P213, P313 Additional analog pulse for X, Y, Z axes

Additional analog pulse to make up for the leadscrew backlash when reversing movement direction.

It is given by an integer between 0 and 255.

Value 0 = There is no additional analog pulse.
Value 1 = 2.5 mV.
Value 10 = 25.0 mV. (10 x 2.5)
Value 255 = 637.5 mV. (255 x 2.5)

Every time the axis moving direction is reversed, the CNC will apply the corresponding analog voltage for that axis plus the additional analog pulse indicated in this parameter during 40 milliseconds.

When the feedback device is a rotary encoder, this parameter must be set to "0".

5.8.2 LEADSCREW ERROR

There is a 30-point leadscrew compensation table for each axis.

For each point, it is required to define the position of the axis and the amount of error at that point.

To access these tables, press the following keystroke sequence:

[AUX]	(SPECIAL FUNCTIONS)
[5]	(AUXILIARY MODES)
[1]	(SPECIAL MODES)
[0] [1] [0] [1]	(ACCESS CODE)
[3]	(LEADSCREW ERROR COMPENSATION)

The operator may view the following or previous pages by using the up and down arrow keys.

To view a particular parameter, key in its number and press **[RECALL]**. The CNC will show the page corresponding to that parameter.

To clear the table by setting all the parameters to 0, key in the following sequence: **[F] [S] [P] [ENTER]**.

Each parameter pair of this table represents:

Even parameter The position of the error point on the leadscrew. This position is referred to Machine Reference Zero (home).

Value range: ±8388.607 millimeters
±330.2599 inches

Odd parameter The amount of leadscrew error at that point.

Value range: ±32.766 millimeters
±1.2900 inches

When defining the compensation points on the table, the following rules must be observed:

* The even parameters are ordered according to their position along the axis. The first pair of parameters (P0 or P60) must be set for the most negative (least positive) point of the axis to be compensated.

* If all 30 points of the table are not required, set the unused ones to 0.

* For those sections outside the compensation area, the CNC will apply the compensation defined for the nearest point.

* The Machine Reference Zero point (home) must be set with an error of 0.

* The maximum difference between the error values of two consecutive compensation points must be within: ±0.127 mm (±0.0050 inches)

* The inclination of the error graph between two consecutive points cannot be greater than 3%.

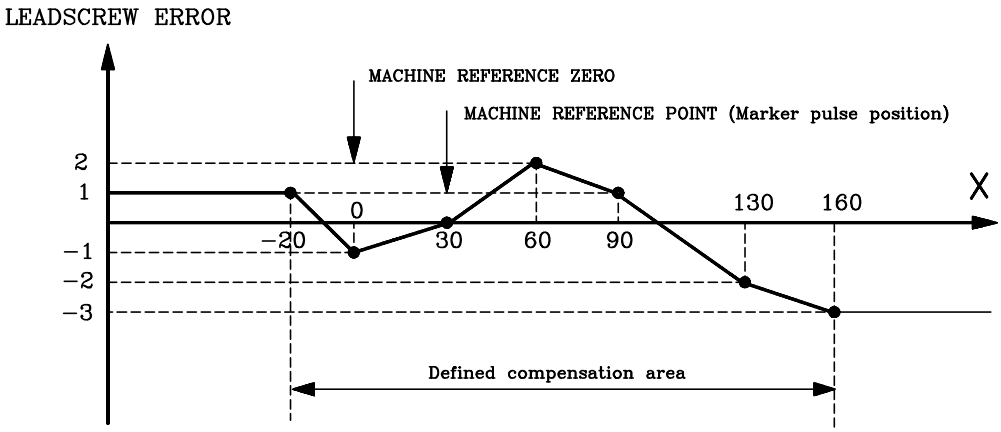
Examples: If the distance between two consecutive points is 3 mm. the maximum difference of their relevant error values can be 0.090 mm.
If the error difference between two consecutive points is the maximum (0.127mm), the distance between them cannot be smaller than 4.233mm.

To EDIT a parameter, key in its number, press **[=]**, key in the desired value and press **[ENTER]** so the new value is entered on the table.

Remember to press **[RESET]** or power the CNC off and back on once the machine parameters have been set in order for the CNC to assume their new values.

Programming example:

An X axis leadscrew is to be compensated according to the following graph in the section between X-20 and X160:



Considering that the machine reference **point** has a value of X30 (meaning that it is located 30mm from the Machine Reference **Zero**), the leadscrew error compensation parameters will be defined as follows:

P000 = X -20.000		P001 = X 0.001	
P002 = X 0.000		P003 = X -0.001	
P004 = X 30.000		P005 = X 0.000	
P006 = X 60.000		P007 = X 0.002	
P008 = X 90.000		P009 = X 0.001	
P010 = X 130.000		P011 = X -0.002	
P012 = X 160.000		P013 = X -0.003	
P014 = X 0.000		P015 = X 0.000	
P016 = X 0.000		P017 = X 0.000	
"	"	"	"
"	"	"	"
P056 = X 0.000		P057 = X 0.000	
P058 = X 0.000		P059 = X 0.000	

P606(8), P606(7), P606(6) Leadscrew error compensation for X, Y, Z

They indicate whether the CNC must apply leadscrew error compensation on the corresponding axis or not.

0 = Leadscrew error compensation **not** applied.
1 = Leadscrew error compensation **applied**.

5.9 SPECIAL MACHINE PARAMETERS

P609(1) Machine travels over 8388.607 mm (330.2599 inches)

This parameter must only be set for those machine having one or more axes with a travel greater than **8388.607 mm (330.2599 inches).**

This parameter **affects both axes** even when one of them might not require this extended travel.

Possible values:

0 = Machine with **normal** axis travel **within** 8388.607 mm (330.2599 inches).
1 = Machine with **extended** axis travel **over** 8388.607 mm (330.2599 inches)

When setting this parameter to "1", the following items must be considered:

* The minimum display resolution for both axes will now be: 0.01mm or 0.001 inch.

* The programming format will now be: ±5.2 in mm and ±4.3 in inches.

* The minimum moving distance will now be: ±0.01mm and ±0.001 inch. The maximum moving distance will be: ±83886.07mm and ±3302.599 inches.

* The tool table format will be affected the same way:

  R,L ±4.2 in mm or ±3.3 in inches. Minimum value: ±0.01mm and ±0.001 inch. Maximum value: ±9999.99mm and ±393.699 inches.

  I,K ±3.2 in mm and ±2.3 in inches. Minimum value: ±0.01mm and ±0.001 inch. Maximum value: ±327.66 mm and ±12.900 inches.

* The integer values assigned to machine parameters P103, P203 and P303 for axis resolution now acquire new units:

  1 = 0.01 mm or 0.001 inch resolution.
  2 = 0.02 mm or 0.002 inch resolution.
  5 = 0.05 mm or 0.005 inch resolution.
  10 = 0.10 mm or 0.010 inch resolution.

* To calculate K1 and K2 and the Feed-forward gain, the following error is now expressed in 0.01 mm units (not microns) and 0.001 inch units (instead of 0.0001 inch as before).

  The maximum amount of following error permissible is now: 320mm

  That is to say that K1 and K2 gains (parameters P114, P214, P314, P116, P216, P316) must be given in "mV/0.01mm (mV/0.001 inches).

* Machine parameters P115, P215 and P315 for gain break-point are now expressed in 0.01 units (not microns) and 0.001 inch units (instead of 0.0001 inch as before).

* Machine parameters P109, P209, P309 (leadscrew backlash) and P118, P218, P318 (in-position zone) will also be given in 0.01 mm and 0.001 inch units.

  P118 = 100 means an in-position zone of 1mm (or 0.1 inch) for the X axis.

* Machine parameters P112, P212, P312, P810, P811, P812 (homing feedrate) and P801 (unidirectional approach) will also be expressed in 0.01mm/min. or 0.001 inch/min.

  P112 = 10000 assigns a feedrate of 100m/min.

Examples to calculate resolution with P609(1)=1:

Example 1: **Resolution in "mm" with square-wave encoder**

We want to obtain a 0.01mm resolution with a square-wave encoder mounted on the X axis whose leadscrew has a 5mm/turn pitch.

Since the multiplying factor applied by the CNC may be either x2 or x4 (depending on machine parameter setting). The resulting encoder line count will be:

$$\text{Number of pulses} = \frac{\text{Leadscrew pitch}}{\text{Multiplying Factor x Resolution}}$$

For a factor of x4:

$$\text{Number of pulses} = \frac{5\text{ mm}}{4 \times 0.01\text{m}} = 125 \text{ pulses/rev.}$$

P103= 1 P604(4)=0 P106=0 P604(8)=0

For a factor of x2:

$$\text{Number of pulses} = \frac{5\text{mm}}{2 \times 0.01\text{ mm}} = 250 \text{ pulses/rev.}$$

P103= 1 P604(4)=0 P106=0 P604(8)=1

Example 2: **Resolution in "inches" with square-wave encoder**

We would like to obtain a 0.001 inch resolution with a square-wave encoder mounted on to the X axis which has a 4-pitch leadscrew (4 turns per inch or 0.25 inch/turn).

Since the CNC always applies a multiplying factor of either x2 or x4 (selected by machine parameter), the required encoder line count (pulses per rev) in each case will be:

$$\text{Number of pulses} = \frac{\text{Leadscrew pitch}}{\text{Multiplying factor x Resolution}}$$

With a x4 factor:

$$\text{Number of pulses} = \frac{0.25}{4 \times 0.001} = \textbf{62.5*} \text{ pulses/turn}$$

*** A gear reduction will be required to achieve this line count per turn.**

P103= 1 P604(4)=1 P106=0 P604(8)=1

With a x2 factor:

$$\text{Number of pulses} = \frac{0.25}{2 \times 0.001} = 125 \text{ pulses/turn}$$

P103= 1 P604(4)=1 P106=0 P604(8)=1

P617(6) Axis resolution of 0.0001mm (tenth of a micron) or 0.00001 inch (10 millionths)

This parameter must only be set for those machine having one or more axes requiring this kind of resolution.

This parameter **affects both axes** even when one of them might not require this kind of resolution.

Possible values:

0 = Machine with **normal** axes with 0.001 mm or 0.0001 inch minimum resolution.

1 = Machine with **special** 0.0001mm or 0.00001 inch resolution.

When setting this parameter to “1”, the following items must be considered:

* The programming format will now be: ±3.4 in mm and ±2.5 in inches.

* The minimum moving distance will now be: ±0.0001mm and ±0.00001 inch. The maximum moving distance will be: ±838.8607mm and ±33.02599 inches.

* The tool table format will be affected the same way:

  R,L ±2.4 in mm or ±1.5 in inches. Minimum value: ±0.0001mm and ±0.00001 inch. Maximum value: ±99.9999mm and ±3.93699 inches.

  I,K ±1.4 in mm and ±0.5 in inches. Minimum value: ±0.0001mm and ±0.00001 inch. Maximum value: ±3.2766 mm and ±0.12900 inches.

* The integer values assigned to machine parameters P103, P203 and P303 for axis resolution now acquire new units:

  1 = 0.0001 mm, 0.00001 inch resolution.
  2 = 0.0002 mm, 0.00002 inch resolution.
  5 = 0.0005 mm, 0.00005 inch resolution.
  10 = 0.0010 mm, 0.00010 inch resolution.

* When calculating K1, K2 gains and the Feed-forward gain, the following error is now expressed in 0.0001 mm units (not microns) and 0.00001 inch units (instead of 0.0001 inch as before). The maximum amount of following error allowed is now: 3.20mm

  That is to say that K1 and K2 gains (parameters P114, P214, P314, P116, P216, P316) must be given in "mV/0.0001mm (mV/0.00001 inches).

* Machine parameters P115, P215, P315 for gain break-point are now expressed in 0.0001 units (not microns) and 0.00001 inch units (instead of 0.0001 inch as before).

* Machine parameters P109, P209, P309 (leadscrew backlash) and P118, P218, P318 (in-position zone) will also be given in 0.0001 mm and 0.00001 inch units.

  P118 = 100 means an in-position zone of 0.01mm (or 0.001 inch) for the X axis.

* Machine parameters P112, P212, P312, P810, P811, P812 (homing feedrate) and P801 (unidirectional approach feedrate) will also be expressed in 0.0001mm and 0.00001 inch units.

P112 = 10000 Assigns a feedrate of 1m/min.

Examples to calculate resolution with P617(6)=1:

Example 1: **Resolution in "mm" with square-wave encoder**

We want to obtain a 0.0001mm resolution with a square-wave encoder mounted on the X axis whose leadscrew has a 5mm/turn pitch.

Since the multiplying factor applied by the CNC may be either x2 or x4 (depending on machine parameter setting). The resulting encoder line count will be:

$$\text{Number of pulses} = \frac{\text{Leadscrew pitch}}{\text{Multiplying Factor x Resolution}}$$

For a factor of x4:

$$\text{Number of pulses} = \frac{5\text{ mm}}{4 \times 0.0001\text{m}} = 12500 \text{ pulses/rev.}$$

P103= 1 P604(4)=0 P106=0 P604(8)=0

For a factor of x2:

$$\text{Number of pulses} = \frac{5\text{mm}}{2 \times 0.0001\text{ mm}} = 25000 \text{ pulses/rev.}$$

P103= 1 P604(4)=0 P106=0 P604(8)=1

Example 2: **Resolution in "inches" with square-wave encoder**

We would like to obtain a 0.001 inch resolution with a square-wave encoder mounted on to the X axis which has a 4-pitch leadscrew (4 turns per inch or 0.25 inch/turn).

Since the CNC always applies a multiplying factor of either x2 or x4 (selected by machine parameter), the required encoder line count (pulses per rev) in each case will be:

$$\text{Number of pulses} = \frac{\text{Leadscrew pitch}}{\text{Multiplying factor x Resolution}}$$

With a x4 factor:

$$\text{Number of pulses} = \frac{0.25}{4 \times 0.00001} = 6250 \text{ pulses/turn}$$

P103= 1 P604(4)=1 P106=0 P604(8)=1

With a x2 factor:

$$\text{Number of pulses} = \frac{0.25}{2 \times 0.00001} = 12500 \text{ pulses/turn}$$

P103= 1 P604(4)=1 P106=0 P604(8)=1

P908, P909 Collision zone between Y, Z

These parameters are used mainly on boring mills and horizontal machining centers to define a possible collision zone between the Y and Z axes.

Each one of them indicates the position value where the collision zone begins.

P908 = Y axis position value.
P909 = Z axis position value.

These position values are absolute and referred to the machine reference zero (home).

Possible values: ± 8388.607 millimeters.
± 330.2599 inches.

If one of the axes gets into this zone, the CNC will prevent the other one from getting into it.

6. *SPINDLE MACHINE PARAMETERS*

6.1 PARAMETERS RELATED TO SPINDLE SPEED RANGE CHANGE

The section on "Spindle range change" in the chapter on "concepts" of this manual describes how these parameters may be used.

P7, P8, P9, P10 Maximum spindle speed for 1st, 2nd, 3rd and 4th RANGE.

They indicate the maximum spindle speed assigned to each range.

It is given in revolutions per minute and they accept any integer value between 0 and 9999.

The value assigned to P7 must correspond to the lowest range and that of P10 to the highest range. When not all the ranges are being used, assign the lowest range to P7 and set the unused ones to the highest speed value.

P601(6) Residual analog voltage (S) during spindle range change.

It determines whether the CNC must generate a residual analog voltage (S) during a range change.

Possible values:

0 = **No** residual analog voltage is generated.
1 = A residual analog voltage **is** generated.

P706 Value of the residual analog voltage (S).

It indicates the value of the residual analog voltage (S) for the spindle speed range change.

It is given by an integer between 1 and 255.

Value of 1 = 2.5 mV.
Value of 10 = 25.0 mV. (10 x 2.5)
Value of 255 = 637.5 mV. (255 x 2.5)

P707 Oscillation period during a spindle range change.

It indicates the oscillation time period during a spindle range change.

It is given by an integer between 0 and 255.

Value of 0 = Continuous movement in one direction.
Value of 1 = Continuous movement in the other direction.
Value of 2 = 20 msec. oscillation period.
Value of 10 = 100 msec. oscillation period.
Value of 255 = 2550 msec. oscillation period.

6.2 PARAMETERS FOR ANALOG SPINDLE SPEED OUTPUT

The section on "Spindle" in the chapter on "concepts" of this manual describes how these parameters may be used.

P601(4) Sign of the spindle analog output.

It determines the sign of the spindle analog output (S). If correct, leave it as is; if not, change it.

Possible values: “0” and “1”.

P610(4) Unipolar or bipolar spindle analog output.

It indicates the type of spindle analog output.

If the analog output is BIPOLAR, the CNC will generate a positive analog voltage (0 to +10V) to turn the spindle clockwise (M03) and a negative analog voltage (0 to -10V) to turn the spindle counter-clockwise (M04).

If the analog output is UNIPOLAR, the CNC will generate a positive analog voltage (0 to +10V) for either turning direction.

0 = The output must be BIPOLAR.
1 = The output must be UNIPOLAR.

Bear in mind that with machine parameter P601(4) it is possible to change the sign of the analog output and, therefore, the spindle turning direction.

P609(4) Any spindle speed change generates an S STROBE

It indicates whether or not a 200msec. S STROBE (pulse) is output at pin 3 of connector I/O1, every time a new spindle speed is selected.

0 = No S STROBE is generated.
1 = An S STROBE is generated.

6.3 PARAMETERS FOR SPINDLE SPEED OUTPUT IN BCD

The section on "Spindle" in the chapter on "concepts" of this manual describes how these parameters may be used.

P601(3) Spindle speed output in 2-digit BCD code.

It indicates whether there is a 2-digit BCD coded spindle speed output or not. If not, the CNC will output an analog voltage for the spindle.

0 = **No 2-digit BCD** coded output is used for spindle speed.
1 = A 2-digit BCD coded output is used for spindle speed.

If this parameter is set to “1”, the CNC will issue the value corresponding to the programmed spindle speed via the BCD outputs which are pins 20 thru 27 of the I/O 1 connector. It will also output an S STROBE pulse at pin 3 of connector I/O 1.

The chart below shows the BCD code corresponding to the programmable spindle speed values:

Programmed S	S BCD	Programmed S	S BCD	Programmed S	S BCD	Programmed S	S BCD
0	S 00	25-27	S 48	200-223	S 66	1600-1799	S 84
1	S 20	28-31	S 49	224-249	S 67	1800-1999	S 85
2	S 26	32-35	S 50	250-279	S 68	2000-2239	S 86
3	S 29	36-39	S 51	280-314	S 69	2240-2499	S 87
4	S 32	40-44	S 52	315-354	S 70	2500-2799	S 88
5	S 34	45-49	S 53	355-399	S 71	2800-3149	S 89
6	S 35	50-55	S 54	400-449	S 72	3150-3549	S 90
7	S 36	56-62	S 55	450-499	S 73	3550-3999	S 91
8	S 38	63-70	S 56	500-559	S 74	4000-4499	S 92
9	S 39	71-79	S 57	560-629	S 75	4500-4999	S 93
10-11	S 40	80-89	S 58	630-709	S 76	5000-5599	S 94
12	S 41	90-99	S 59	710-799	S 77	5600-6299	S 95
13	S 42	100-111	S 60	800-899	S 78	6300-7099	S 96
14-15	S 43	112-124	S 61	900-999	S 79	7100-7999	S 97
16-17	S 44	125-139	S 62	1000-1119	S 80	8000-8999	S 98
18-19	S 45	140-159	S 63	1120-1249	S 81	9000-9999	S 99
20-22	S 46	160-179	S 64	1250-1399	S 82		
23-24	S 47	180-199	S 65	1400-1599	S 83		

If a value greater than 9999 is programmed, the CNC will assume the spindle speed corresponding to 9999.

P601(2) 4-digit BCD coded spindle speed output

It indicates whether there is a 4-digit BCD coded spindle speed output or not. If not, the CNC will output an analog voltage for the spindle.

0 = **No 4-digit BCD** coded output is used for spindle speed.
1 = A 4-digit BCD coded output is used for spindle speed.

If this parameter is set to "1", the CNC will issue the value corresponding to the programmed spindle speed via the BCD outputs which are pins 20 thru 27 of the I/O 1 connector.

The CNC will output the value corresponding to the programmed S in two stages with a 100 msec. delay between them. It will also output an S STROBE pulse at pin 3 of connector I/O 1.

Pin	1st stage	2nd stage
20 21 22 23	Thousands	Tens
24 25 26 27	Hundreds	Units

7. CONCEPTS

Atention:

It is recommended to save the CNC machine parameters onto a peripheral device or PC to avoid losing them by mistake or any malfunction.

7.1 FEEDBACK SYSTEMS

The feedback inputs of this CNC are:

Connector A1.

They are used to connect the feedback for the X axis.

It admits sine-wave and double-ended (differential) square-wave signals which must be properly indicated by machine parameters “P106" and the two dip-switches located under the feedback input.

Connector A2.

They are used to connect the feedback for the Y axis.

It admits sine-wave and double-ended (differential) square-wave signals which must be properly indicated by machine parameters “P206" and the two dip-switches located under the feedback input.

Connector A3.

They are used to connect the feedback for the Z axis.

It admits sine-wave and double-ended (differential) square-wave signals which must be properly indicated by machine parameters “P306" and the two dip-switches located under the feedback input.

Connector A4.

It is used to connect the electronic handwheel associated with the Y axis.

Set the two dip-switches, located under the feedback input, accordingly.

Connector A5.

It is used to connect the electronic handwheel associated with the Z axis.

Set the two dip-switches, located under the feedback input, accordingly.

Connector A6.

It is used to connect the electronic handwheel associated with the X axis.

7.1.1 COUNTING FREQUENCY LIMITS

Sine-wave signals

The maximum counting frequency for sine-wave feedback signals is 25KHz (25,000 pulses/sec.).

Therefore, the maximum feedrate for each axis will depend upon the selected resolution (machine parameters "P103, P203, P303") and the period of the feedback signal being used.

Example 1:

When using a linear scale with a 20µm, the maximum feedrate for an axis with 1µm resolution will be:

20 µm/pulse x 25,000 pulses/sec = 500 mm/sec = 30 m/min.

Square-wave signals

The maximum counting frequency for differential square-wave signals is 200 KHz (200,000 pulses/sec.), with a 450nsec. separation between A and B flanks (that is 90º ±20º phase shift).

Therefore, the maximum feedrate for each linear axis will depend upon the selected resolution (machine parameters "P103, P203, P303") and the period of the feedback signal being used.

When using FAGOR linear scales, the maximum feedrate is limited by its own characteristics to 60m/min (2362 inches/min).

When using FAGOR rotary encoders, the limitation is set by the maximum number of pulses delivered by the encoder; which is 200KHz.

7.2 MOVEMENT BY ELECTRONIC HANDWHEEL

This CNC may have up to 3 electronic handwheels associated to their axes (one per each axis, X, Y, Z). They can be set by machine parameter.

These handwheels will be active in the JOG mode when the feedrate override switch (F.O.S) on the operator panel is set at any position indicated by the symbol.

The handwheel positions available at the F.O.S. are 1, 10 and 100 which indicate the multiplication factor applied to the handwheel pulses.

This way, the actual axis movement results from multiplying the handwheel pulses by the selected factor.

Those units correspond to the display units being selected.

Example:

Handwheel line count: 250 pulses per turn

F.O.S. position	Distance moved per turn
1	0.250 mm or 0.0250 inch
10	2.500 mm or 0.2500 inch
100	25.000 mm or 2.5000 inches

The feedback connectors to be used for each axis and each handwheel are:

A6 for the X axis handwheel
A4 for the Y axis handwheel
A5 for the Z axis handwheel

When using only the FAGOR 100P handwheel with axis selector button, it must be connected to A6.

7.3 AXIS RESOLUTION

The CNC has a series of machine parameters to set the resolution of each axis.

The resolution used on each axis indicates the minimum variation distinguishable by the feedback device. It is given in microns or 0.0001 inch units.

The machine parameters used to define the axis resolution are the following:

P103, P203, P303	They set the counting resolution for each axis.
P604(4), P604(3), P604(2)	They set the measuring units for each axis feedback signal (mm or inches).
P106, P206, P306	They set the type of feedback signal being used (square-wave or sine-wave) for each axis.
P604(8), P604(7), P604(6)	They indicate the multiplying factor, x2 or x4 to be applied to the feedback signals of each axis.
P622(1), P622(2), P622(3)	They indicate the special multiplying factor to be applied to the sine-wave feedback signals of each axis (besides the normal x5).

Example 1: **Resolution in "mm" with square-wave encoder**

We want to obtain a 2µm resolution with a square-wave encoder mounted on the X axis whose leadscrew has a 5mm/turn pitch.

Since the multiplying factor applied by the CNC may be either x2 or x4 (depending on machine parameter setting). The resulting encoder line count will be:

$$\text{Number of pulses} = \frac{\text{Leadscrew pitch}}{\text{Multiplying Factor x Resolution}}$$

For a factor of x4:

$$\text{Number of pulses} = \frac{5000\ \mu m}{4 \times 2\ \mu m} = 625 \text{ pulses/rev.}$$

P103= 2 P604(4)=0 P106=0 P604(8)=0

For a factor of x2:

$$\text{Number of pulses} = \frac{5000\ \mu m}{2 \times 2\ \mu m} = 1250 \text{ pulses/rev.}$$

P103= 2 P604(4)=0 P106=0 P604(8)=1

If a FAGOR encoder is chosen, its pulse output frequency is limited to 200KHz (although the CNC admits square-wave pulses with a frequency of up to 200KHz). Therefore, the maximum feedrate for this axis will be:

When using a x4 multiplying factor:

$$\text{Max. Feed} = \frac{200{,}000 \text{ pulses/sec.}}{625 \text{ pulses/rev.}} \times 5 \text{ mm/rev.} = 1600 \text{ mm/sec.} = 96 \text{ m/min.}$$

When using a x2 multiplying factor:

$$\text{Max. Feed} = \frac{200{,}000 \text{ pulses/sec.}}{1250 \text{ pulses/rev.}} \times 5 \text{ mm/rev.} = 800 \text{ mm/sec.} = 48 \text{ m/min.}$$

Example 2: **Resolution in "mm" with sine-wave encoder**

We would like to get a 2µm resolution with a sine-wave encoder mounted on to the X axis which has a 5mm/turn leadscrew pitch.

We have the following options:

P622(1)=0		
P604(8)	**Resolution**	**P103**
=1 (x2)	2 microns	10
=0 (x4)	2 microns	10

P622(1)=1		
P604(8)	**Resolution**	**P103**
=1 (x2)	2 microns	2
=0 (x4)	2 microns	2

Since the CNC always applies a x5 multiplying factor to the sine-wave feedback signals, we will need an encoder:

$$\text{Nº of pulses} = \frac{\text{Leadscrew pitch}}{5 \times \text{Multiplying factor} \times \text{Resolution}}$$

For P604(8)=1 (x2)

$$\text{Nº pulses} = \frac{5000\mu m/turn}{5 \times 2 \times 2\mu m/pulse} = 250 \text{ pulses/turn}$$

Therefore:

If P622(1)=0 => P604(4)=0 P106=1 P604(8)=1 P103= 10
If P622(1)=1 => P604(4)=0 P106=1 P604(8)=1 P103= 2

For P604(8)=0 (x4)

$$\text{Nº pulses} = \frac{5000\mu m/turn}{5 \times 4 \times 2\mu m/pulse} = 125 \text{ pulses/turn}$$

Therefore:

If P622(1)=0 => P604(4)=0 P106=1 P604(8)=0 P103= 5
If P622(1)=1 => P604(4)=0 P106=1 P604(8)=0 P103= 2

Even when choosing a FAGOR encoder which outputs up to 200KHz, the actual usable frequency is this time limited by the CNC to 25KHz for sine-wave signals. Therefore, the maximum feedrate for this example will be:

$$\text{Max. Feed} = \frac{25{,}000 \text{ pulses/sec.}}{125 \text{ pulses/rev.}} \times 5 \text{ mm/rev.} = 1000 \text{ mm/sec.} = 60 \text{ m/min.}$$

and 30m/min for 250-line encoder.

Example 3: **Resolution in "mm" with square-wave linear scale**

Considering that the CNC applies either a x2 or x4 multiplying factor (set by machine parameter), a linear scale must be chosen whose pitch is 2 or 4 times the desired resolution.

When using FAGOR linear transducers (scales) with 20µm pitch, the following resolutions may be obtained: 5µm (20/4), 10µm (20/2).

Thus:

Scale pitch	P103	P604(4)	P106	P604(8)
20µm	5	0	0	0
20µm	10	0	0	1

Since the counting frequency of the CNC is limited to 200KHz for square-wave signals, the maximum feedrate obtainable with a 20µm-pitch scale is:

Max. Feed = 20 µm/pulse x 200,000 pulses/sec. = 4000 mm/sec. = **240 m/min.**

However, if FAGOR linear scales are used, the maximum feedrate is limited (by the scales) to 60m/min. (2362 inches/min.)

Example 4: **Resolution in "mm" with sine-wave linear scales**

A sine-wave linear scale is being used with a 20µm pitch and 1µm resolution.

There are the following options:

P622(1)=0		
P604(8)	**Resolution**	**P103**
=0 (x4)	1 micron	5

P622(1)=1		
P604(8)	**Resolution**	**P103**
=0 (x4)	1 micron	1

Therefore:

If P622(1)=0 => P604(4)=0 P106=1 P604(8)=0 P103= 5
If P622(1)=1 => P604(4)=0 P106=1 P604(8)=0 P103= 1

The CNC's counting frequency is limited to 25KHz (30m/min) for sine-wave signals.

However, if FAGOR linear scales are used, the maximum feedrate is limited (by the scales) to 60m/min. (2362 inches/min.)

Example 5: **Resolution in "inches" with square-wave encoder**

We would like to obtain a 0.0001 inch resolution with a square-wave encoder mounted on to the X axis which has a 4-pitch leadscrew (4 turns per inch or 0.25 inch/turn).

Since the CNC always applies a multiplying factor of either x2 or x4 (selected by machine parameter), the required encoder line count (pulses per rev) in each case will be:

$$\text{Number of pulses} = \frac{\text{Leadscrew pitch}}{\text{Multiplying factor x Resolution}}$$

With a x4 factor:

$$\text{Number of pulses} = \frac{0.25}{4 \times 0.0001} = 625 \text{ pulses/turn}$$

P103= 1 P604(4)=1 P106=0 P604(8)=0

With a x2 factor:

$$\text{Number of pulses} = \frac{0.25}{2 \times 0.0001} = 1250 \text{ pulses/turn}$$

P103= 1 P604(4)=1 P106=0 P604(8)=1

If a FAGOR encoder is used, the counting frequency is limited to 200KHz by the scale (the CNC admits up to 200KHz for square-wave signals). Therefore, the maximum feedrate for this axis will be:

For x4 multiplying factor:

$$\text{Max. Feed} = \frac{200{,}000 \text{ pulses/sec.}}{625 \text{ pulses/rev}} \times 0.25 \text{ inch/rev} = 80 \text{ inches/sec} = 4800 \text{ inch/min.}$$

For x2 multiplying factor:

$$\text{Max. Feed} = \frac{200{,}000 \text{ pulses/sec.}}{1250 \text{ pulses/rev}} \times 0.25 \text{ inch/rev} = 40 \text{ inches/sec} = 2400 \text{ inch/min.}$$

Example 6: Resolution in "inches" with sine-wave encoder

We would like to get a 0.0001 inch resolution with a sine-wave encoder mounted on to the X axis which has a 4-pitch leadscrew (0.25 inch/turn).

We have the following options:

P622(1)=0		
P604(8)	**Resolution**	**P103**
=1 (x2)	0.0001 inch	5
=0 (x4)	0.0001 inch	5

P622(1)=1		
P604(8)	**Resolution**	**P103**
=1 (x2)	0.0001 inch	1
=0 (x4)	0.0001 inch	1

Since the CNC always applies a x5 multiplying factor to the sine-wave feedback signals, we will need an encoder:

$$\text{Nº of pulses} = \frac{\text{Leadscrew pitch}}{5 \times \text{Multiplying factor} \times \text{Resolution}}$$

For P604(8)=1 (x2)

$$\text{Nº pulses} = \frac{0.25 \text{ inch/turn}}{5 \times 2 \times 0.0001 \text{ inch/pulse}} = 250 \text{ pulses/turn}$$

Therefore:

If P622(1)=0 => P604(4)=0 P106=1 P604(8)=1 P103= 5
If P622(1)=1 => P604(4)=0 P106=1 P604(8)=1 P103= 1

For P604(8)=0 (x4)

$$\text{Nº pulses} = \frac{0.25 \text{ inch/turn}}{5 \times 4 \times 0.0001 \text{ inch/pulse}} = 125 \text{ pulses/turn}$$

Therefore:

If P622(1)=0 => P604(4)=0 P106=1 P604(8)=0 P103= 5
If P622(1)=1 => P604(4)=0 P106=1 P604(8)=0 P103= 1

Even when choosing a FAGOR encoder which outputs up to 200KHz, the actual usable frequency is this time limited by the CNC to 25KHz for sine-wave signals. Therefore, the maximum feedrate for this example will be:

$$\text{Max. Feed} = \frac{25{,}000 \text{ pul./sec.}}{125 \text{ pul./rev.}} \times 0.25 \text{ inch/rev.} = 500 \text{ inch/sec.} = 3000 \text{ inch/min.}$$

and 1500 inch/min for 250-line encoder.

7.4 ADJUSTMENT OF THE AXES

In order to make this adjustment it is necessary to have the feedback systems for all the axes connected to the CNC.

Before starting the adjustment of the axes, it is a good idea to move them close to the middle of their travels placing the travel-limit switches (controlled by the electrical cabinet) close to these points in order to avoid any damage to the machine.

Verify that the axes are not CONTINUOUSLY CONTROLLED (that their enable signal is cancelled once the axis is in position). To do this, machine parameters P105, P205 and P305 must be set to “0”.

Also, make sure that the CNC has been set to provide a delay between the axes enable and their analog voltage output. To do this, machine parameters P104, P204 and P304 must be set to “1”.

After the machine parameters for the axes have been properly set, proceed with their adjustment by following these suggestions:

* The axes should be adjusted one at a time.

* Connect the power output of the drive corresponding to the axis being adjusted.

* In the JOG mode, move the axis being adjusted.

  In case of run-away, the CNC will display the relevant following error and the machine parameter corresponding to the SIGN OF THE ANALOG VOLTAGE will have to be changed. Machine parameters P100, P200 and P300.

* If the axis does not run away; but the direction of the move is not the desired one, the machine parameter corresponding to the COUNTING DIRECTION (P101, P201 and P301) will have to be changed as well as that corresponding to the SIGN OF THE ANALOG VOLTAGE (P100, P200 and P300).

* If the counting direction is correct but the axis moves in the opposite direction to the one indicated, change the axis machine parameter corresponding to the moving direction (P102, P202 and P302).

7.4.1 ADJUSTMENT OF THE DRIFT (OFFSET) AND MAXIMUM FEEDRATE

These adjustments are performed on axis servo drives and spindle drives.

Drift adjustment (offset)

This adjustment will be made in two stages:

Preadjustment of the drive offset

* Disconnect the analog voltage input of the drive and short-circuit it with a wire jumper.

* Turn the offset potentiometer of the drive until the voltage on the tacho terminals is 0V. This should be checked on the 200 mV DC scale of the volt-meter.

* Remove the wire jumper mentioned above.

Critical adjustment of the drive offset

* Move the axis continuously back and forth.

  For example; Assign the maximum travel by using BEG X and END X and execute the following commands:

  The X axis will continuously move back and forth from BEGIN to END until is pressed

  When single mode is selected (by means of the key), the key must be pressed at the end of each move.

  While the axis is moving, turn the offset potentiometer of the drive until the amounts of following error obtained in both directions are the same.

Adjustment of the maximum feedrate

It is recommended to adjust the drives so the maximum feedrate is obtained with an analog voltage of 9.5V.

Also, the maximum feedrate must be indicated in the corresponding machine parameter for that axis. Parameter P111, P211, P311.

The maximum feedrate can be calculated from the motor rpm, the gear ratios and the type of leadscrew being used.

Example for the X axis:

A motor can turn at 3000 rpm and it is attached to a 5 pitch leadscrew (1/5 inch/turn). Therefore, the maximum feedrate to be assigned to machine parameter P111 is:

Maximum feedrate (G00) = r.p.m. x leadscrew pitch

P111 = 3000 rev./min. x 1/5 inch/rev. = 600 inches/min.

To make this adjustment, it is recommended to set P110 and P111 to the same value.

Also move the axis continuously back and forth.

For example; Assign the maximum travel by using BEG X and END X and execute the following commands:

The X axis will continuously move back and forth from BEGIN to END until is pressed

When single mode is selected (by means of the

key), the

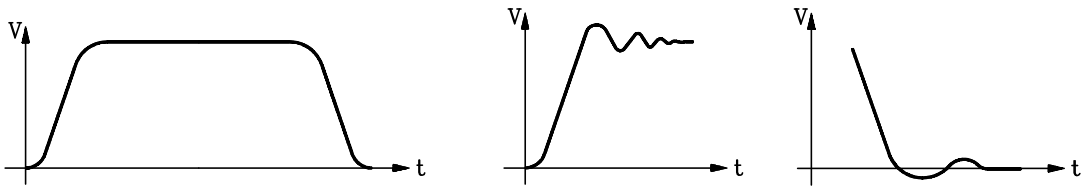
key must be pressed at the end of each move.

While the axis is moving, measure the analog voltage coming out of the CNC towards the servo drive and adjust the gain potentiometer **at the servo drive (never at the CNC)** until this analog voltage reaches 9.5V.

7.4.2 GAIN ADJUSTMENT

It is necessary to properly adjust the different gains for each axis in order to optimize the response of the whole system to the programmed movements.

It is recommended to use an oscilloscope in order to obtain a finer adjustment of the axes by monitoring the signals provided by the tacho. The diagram on the left corresponds to the ideal signal shape and the other ones to an unstable start-up and brake-down.



The CNC has a series of machine parameters which permit adjusting the proportional gain for each axis. These parameters are:

PROPORTIONAL GAIN K1.

Defined by parameters: P114, P214, P314.

PROPORTIONAL GAIN K2.

Defined by parameters: P116, P216, P316.

Value of the GAIN BREAK POINT

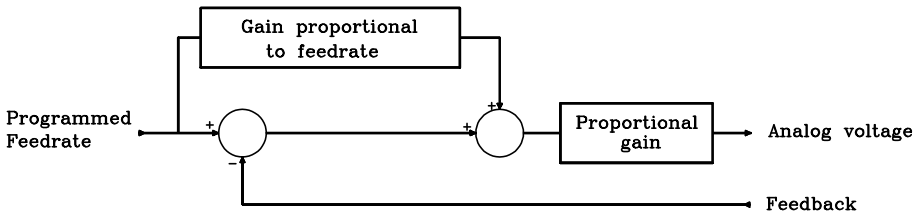
Defined by parameters: P115, P215, P315.

FEED-FORWARD GAIN or gain proportional to the feedrate.

Defined by parameters: P732, P733, P734.

The parameters corresponding to the proportional gain K1 and K2 as well as for the gain break point allow adjusting the Proportional Gain for the axis.

The parameter for the Feed-Forward gain (proportional to feedrate) will be used when acceleration/deceleration control is being applied onto the corresponding axis.



7.4.3 PROPORTIONAL GAIN ADJUSTMENT

The analog voltage supplied by the CNC to control the axis is, at all times, a function of the amount of following error; that is, the difference between the theoretical position and the real (actual) position of the axis.

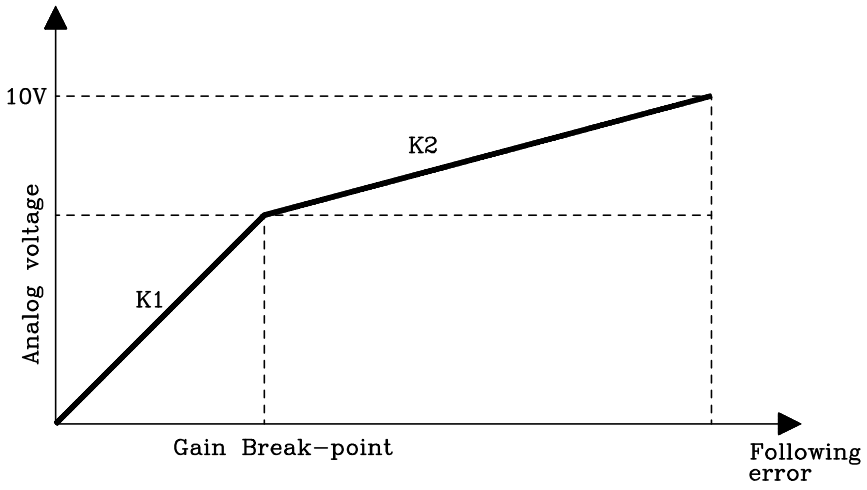
Analog output = Proportional gain “K” x Following Error

On start-up and slow-down, the following error of the axis is very small. Therefore, the proportional gain must be great in order for the axis to respond properly.

On the other hand, once the axis reaches its programmed speed, the following error is maintained practically constant and it is necessary to apply a smaller gain (K) in order to keep the system stable.

The FAGOR 800M CNC offers two proportional gains K1 and K2 to better adjust the system as well as another parameter referred to as Gain Break point which defines the active area for each one of these gains.

The CNC applies the proportional gain K1 whenever the amount of following error for the axis is smaller than the value assigned to the machine parameter corresponding to the gain break-point.



When the amount of following error exceeds the gain break-point value, the CNC applies the K2 value.

Analog = (K1 x Ep) + [K2 x (Following Error - Ep)]

Where “Ep” is the value assigned to the gain break-point and it is given in microns.

When adjusting the proportional gain, it must be borne in mind that:

* When the amount of following error exceeds 32mm (1.2598 inches) the CNC will issue a Following error message for the corresponding axis.

* The amount of following error will decrease as the gain value increases, but the system will tend to be more unstable.

* In practice, most machines seem to respond well to what is called a unity gain (or gain of 1) which represents a following error of 1mm at a feedrate or 1m/minute or a following error of 0.001 inch at a feedrate of 1 inch/min.

  Therefore, this could be used as a practical starting point for the gain calculation described next. After analyzing the behavior of the machine for this gain, its value may be changed in order to optimize it.

7.4.3.1 CALCULATION OF K1, K2 AND GAIN BREAK-POINT

The value of K1 represents the analog voltage corresponding to 1 micron of following error. It is given by an integer between 0 and 255 in such a way that a value of 64 corresponds to an analog voltage of 2.5mV.

Therefore, the K1 value corresponding to a gain of 1 and a maximum motor speed adjusted to 9.5V servo analog input would be given by the following formulae:

$$\text{In metric (FE= 1mm for F=1m/min): } K1 = \frac{243.2}{\text{Fmax in m/min (P111 for X)}}$$

$$\text{In inches (FE= 0.001inch for F=1inch/min): } K1 = \frac{9575}{\text{Fmax in inch/min (P111 for X)}}$$

For example:

If the top feedrate for an axis is 500 inches/min (P111 for X), the K1 corresponding to a unity gain would be: K1 = 9575/500 = 19.15 and the value assigned to the corresponding parameter would be K1=19.

If the top feedrate for an axis is 20m/min, the K1 corresponding to a unity gain would be: K1 = 243.2/20 = 12.16 and the value assigned to the corresponding parameter would be K1=12.

The amount of following error corresponding to the GAIN BREAK-POINT is given in microns or 0.0001 inch units (by parameter P115 for the X axis, P215 for the Y axis, etc.). It is recommended to set it to a value slightly greater than the following error corresponding to the maximum machining feedrate F0 (P110, P210, P310).

For example:

Let's suppose that K1 has been set for a gain of 1 (not K1=1) and that the maximum machining feedrate is 150 inches/min (P110, P210, P310). At this feedrate, the following error should be about 0.150 inch. Thus, the gain break-point value should be slightly larger than 0.150 inch; for example: P115=0.155 inch.

Or in metric:

Let's suppose that K1 has been set for a gain of 1 (not K1=1) and that the maximum machining feedrate is 5 m/min (P110, P210, P310). At this feedrate, the following error should be about 5 mm. Thus, the gain break-point value should be slightly larger than 5 mm; for example: P115=6 mm.

The machine parameter K2 gain sets the analog voltage for 1 micron of following error being applied from the gain break-point on.

It is also given by an integer between 0 and 255 and it is typically set to a value between 50% and 70% of K1 in order to avoid abrupt analog voltage changes when switching to slow machining feedrates.

To perform a practical axis adjustment at the machine, it is recommended:

1.- Adjust K1 optimizing the axis' response like the signal diagram shown earlier.

Set K1 =K2 or set the gain break-point to a large value (for example: 50000) and run a program moving the axis continuously back and forth.

For example; Assign the maximum travel by using BEG X and END X and execute the following commands:

The X axis will continuously move back and forth from BEGIN to END until is pressed

When single mode is selected (by means of the

key), the key must be pressed at the end of each move.

Adjust the value of K1 until the proper response is obtained.

2.- Set the gain-break point to the correct value.

To do this, run the previous program and watch the amount of following error reached at **maximum** feedrate and assign that value or one slightly larger to the gain break-point.

3.- Once K1 and the gain break-point have been set, change the value of K2 to one between 50% and 70% of K1.

Atention:

Once each axis has been adjusted separately. All interpolating axes should be fine adjusted together in such a way that their following errors for the same feedrate are the same in order to achieve proper interpolations between those axes in the K1 area.

7.4.4 FEED-FORWARD GAIN ADJUSTMENT.

With the Feed-Forward gain it is possible to improve the positioning loop of the axes, thus minimizing the amount of following error. **This gain must be used only when working with ACC/DEC.**

This CNC offers two types of acc/dec.:

Linear

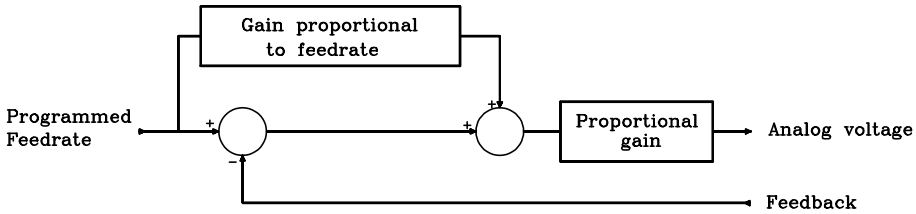
It is mainly applied onto G00 and F00 moves, although it may also be used in G01 moves.

Bell shape:

This type may be used on all kinds of movements, G00, G01, G02, etc. and with any type of feedrate F.

7.4.4.1 CALCULATION OF FEED-FORWARD GAIN

The Feed-Forward gain is proportional to the feedrate and is set by machine parameters P732, P733, P734 which indicate the % of analog voltage that is due to the programmed feedrate.



The value added to the following error is (Kf x F/6) where Kf is the value of Feed-Forward and F is the programmed feedrate.

The CNC will apply the proportional gain (K1 and K2) to the value resulting from adding the following error of the machine plus the value selected by the Feed-Forward.

When the result of the addition is smaller than the value of the gain break-point, the CNC will apply the formula:

Analog = K1 x [Following Error + (Kf x F/6)]

And when the result of the addition is greater than the value of the gain break-point, the CNC will apply the formula:

Analog = (K1 x Ep) + {K2 x [Following Error + (Kf x F/6) - Ep]}

Where “Ep” is the value of the gain break-point.

7.4.5 LEADSCREW ERROR COMPENSATION

With this CNC it is possible to compensate for leadscrew error as well as for its backlash when reversing movement direction.

There is a 30-point leadscrew error compensation table for each axis. To access these tables, press the following keystroke sequence:

[AUX]	(SPECIAL FUNCTIONS)
[5]	(AUXILIARY MODES)
[1]	(SPECIAL MODES)
[0] [1] [0] [1]	(ACCESS CODE)
[3]	(LEADSCREW ERROR COMPENSATION)

The operator may view the following or previous pages by using the up and down arrow keys.

To view a particular parameter, key in its number and press **[RECALL]**. The CNC will show the page corresponding to that parameter.

To clear the table by setting all the parameters to 0, key in the following sequence: **[F] [S] [P] [ENTER]**.

Each table has 30 points; two parameters per point.

X axis table:	P0 through P59.
Y axis table:	P60 through P119.
Z axis table:	P120 through P179.

Each parameter pair of this table represents:

Even parameter The position of the error point on the leadscrew. This position is referred to Machine Reference Zero (home).

Value range: ±8388.607 millimeters
±330.2599 inches

Odd parameter The amount of leadscrew error at that point.

Value range: ±32.766 millimeters
±1.2900 inches

When defining the compensation points on the table, the following rules must be observed:

* The even parameters are ordered according to their position along the axis. The first pair of parameters (P0, P60 or P120) must be set for the most negative (least positive) point of the axis to be compensated.

* If all 30 points of the table are not required, set the unused ones to 0.

* For those sections outside the compensation area, the CNC will apply the compensation defined for the nearest point.

* The Machine Reference Zero point (home) must be set with an error of 0.

* The maximum difference between the error values of two consecutive compensation points must be within: ±0.127 mm (±0.0050 inches)

* The inclination of the error graph between two consecutive points cannot be greater than 3%.

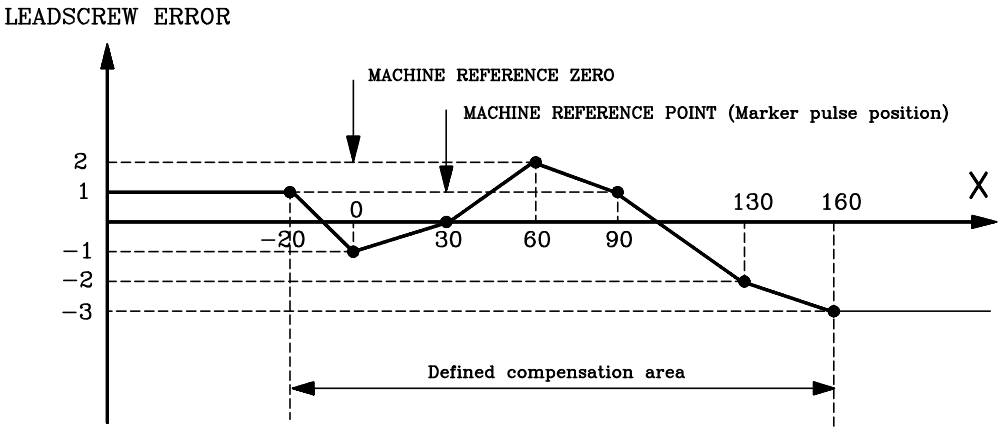
  Examples: If the distance between two consecutive points is 3 mm. the maximum difference of their relevant error values can be 0.090 mm.
  If the error difference between two consecutive points is the maximum (0.127mm), the distance between them cannot be smaller than 4.233mm.

To EDIT a parameter, key in its number, press [=], key in the desired value and press [ENTER] so the new value is entered on the table.

Remember to press RESET or power the CNC off and back on once the machine parameters have been set in order for the CNC to assume their new values.

Programming example:

An X axis leadscrew is to be compensated according to the following graph in the section between X-20 and X160:



Considering that the machine reference **point** has a value of X30 (meaning that it is located 30mm from the Machine Reference **Zero**), the leadscrew error compensation parameters will be defined as follows:

P000	= X -20.000	P001	= X 0.001
P002	= X 0.000	P003	= X -0.001
P004	= X 30.000	P005	= X 0.000
P006	= X 60.000	P007	= X 0.002
P008	= X 90.000	P009	= X 0.001
P010	= X 130.000	P011	= X -0.002
P012	= X 160.000	P013	= X -0.003
P014	= X 0.000	P015	= X 0.000
P016	= X 0.000	P017	= X 0.000
"	"	"	"
"	"	"	"
P056	= X 0.000	P057	= X 0.000
P058	= X 0.000	P059	= X 0.000

7.5 REFERENCE SYSTEMS

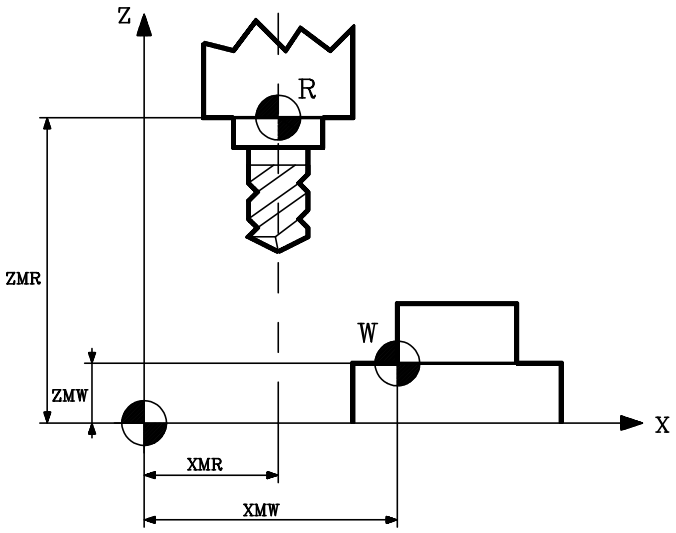
7.5.1 REFERENCE POINTS

A CNC machine must have the following reference points established:

* **Machine Reference Zero** or origin point of the machine. It is set by the machine manufacturer as the origin of the coordinate system of the machine.

* **Part Zero** or origin point for the part. It is the origin point set to program the measurements of the part. It can be chosen freely by the programmer and its reference to the machine reference zero is set by means of a zero offset.

* **Machine Reference Point**. It is the physical location of the marker pulse or reference pulse (Io) used as home to synchronize the whole machine coordinate system. The axis moves to this point when being "homed" and the CNC assumes the reference values set at machine parameter "P119, P219, P319" accordingly.



M	Machine Reference Zero
W	Part Zero
R	Machine Reference Point
XMW, YMW, ZMW, etc.	Part Zero coordinates
XMR, YMR, ZMR, etc.	Machine Reference Point coordinates

7.5.2 MACHINE REFERENCE SEARCH (HOME)

With this CNC it is possible to search home on each axis ([X] [up arrow], [Y] [up arrow], and [Z] [up arrow]) as follows:

1.- The CNC starts moving the axis in the direction set by machine parameter P623(8) for X, P623(7) for Y and P623(6) for Z at the feedrate set by machine parameter P112 for X, P212 for Y, and P312 for Z until the home switch is pressed.

If the selected axis has no home switch (parameters "P602(4), P602(3), P602(2)"), the CNC will consider it to be pressed and it will go on with paragraph 2.

2.- Once the home switch is pressed, the CNC will continue moving the axis at the feedrate selected by machine parameters P810 for X, P811 for Y and P812 for Z until the reference pulse (marker, Io) of the feedback device is found.

Once the home search is completed, the CNC will cancel the selected zero offset and it will display the coordinates of the machine reference point indicated by machine parameter P119 for X, P219 for Y and P319 for Z.

If machine parameter "P611(2) = 1" so it is mandatory to home all the axes on machine power-up, the CNC will behave as follows:

* The CNC assumes as axis feedrate, for JOG and handwheels, the value set by machine parameter "P804".

* A new feedrate value cannot be entered until all the axes of the machine have been referenced (homed).

* The feedrate override may be varied between 0 and 100% by means of the Feedrate Override Switch on the front panel. When selecting the 110% or the 120% position, the CNC only applies 100% unless all the axes have been homed.

* The CNC keeps the "Mandatory Io" output (pin 12 of connector I/O2) high until all the axes have been homed.

  Care must be taken when using this feature so the corresponding bit of the decoded M table is not used (M10 output) since the CNC will activate this output in both cases.

7.5.3 ADJUSTMENT OF THE VALUE CORRESPONDING TO THE MACHINE REFERENCE POINT (HOME)

Before making this adjustment, place the mechanical travel limits (stops) where they are supposed to go.

One of the procedures that can be followed is this:

1.- Define the machine parameters related to home.

Set P602(4), P602(3) and P602(2) to "1" indicating that both axes have home switches.

Set parameters P600(8), P600(7) and P600(6) indicating the type of marker pulse used by the feedback device when searching home. Fagor scales have one negative marker pulse (Io) every 50mm and Fagor rotary encoders have one positive marker pulse per revolution.

Also, set parameters P623(8), P623(7) and P623(6) to indicate the direction the axes must move when searching home.

Besides, set parameters P112, P212 and P312 to indicate the home switch approaching feedrate (1st home searching feedrate) as well as parameters P810, P811and P812 to indicate the marker pulse approaching feedrate (2nd home searching feedrate).

Assign a value of "0" to the machine reference point. Parameters P119, P219 and P319.

2.- Position the axis in the proper area for home search and execute the home search command.

Press **[X]**, **[Y]** or **[Z]** and then, **[up arrow]** followed by

The CNC will carry out the home search and when done, it will assign a value of zero to that point.

3.- Move the axis to the physical location where machine zero point will be (or to a position whose distance to machine zero is known), write down the position value displayed by the CNC at that point.

The value to be assigned to parameter P119, P219 or P319 will be:

Machine coordinate of the measured point - CNC value at that point.

Example for the X axis:

If the point of known dimensions is 230mm from the machine zero and the CNC shows "-123.5mm", the value to be assigned to P119 will be:
"P119" = 230 - (-123.5) = 353.5 mm.

4.- Assign this new value to the machine parameter and press

or power the CNC down and back up in order for the CNC to assume this new value.

5.- Perform a new home search in order for the CNC to assume the correct reference values.

7.5.4 SOFTWARE TRAVEL LIMITS FOR THE AXES

Once the home search has been carried out on all the axes, the soft limits for the CNC have to be established.

This is achieved a single axis at a time and in the following manner:

* Jog the axis in the positive direction to a point close to the travel limit switch keeping a safety distance from it.

* Assign the position value displayed by the CNC to the machine parameter corresponding to the positive software travel limit. Parameter P107, P207, P307.

* Repeat those steps in the negative direction assigning the displayed value to machine parameter corresponding to the negative software travel limit. Parameter P108, P208, P308.

* Once this process is completed, press RESET or turn the CNC off and back on in order for the new values to be assumed by the CNC.

7.5.5 CONSIDERATIONS ABOUT THE MACHINE REFERENCE POINT

* If at the instant the home search is initiated, the home switch is pressed, the axis will withdraw [in the opposite direction to that set by P623(8), P623(7), P623(6)] until releasing the home switch before starting the actual home search.

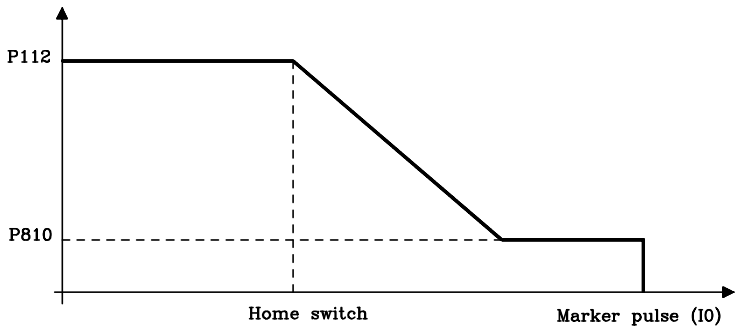
* If the axis is out of the soft travel limits (set by P107-P108, P207-P208, P307-P308), it has to be jogged into the work area (within limits) and, then, positioned at the correct side from home before starting the actual home search.

* Care must be taken when placing the home switch and when setting the home searching feedrates (P112, P212, P312, P810, P811, P812) to prevent any overshooting.

* If the selected axis does not have a home switch [P602(4), P602(3), P602(2)], the CNC will consider it to be pressed and it will only make the marker pulse searching move at the feedrate set by P810, P811, P812 until the marker pulse (Io) from the feedback device is detected; thus completing the home search.

* FAGOR linear transducers (scales) have a negative marker pulse (Io) every 50 mm ["P600(8), P600(7), P600(6)" = 0] and FAGOR rotary encoders output a positive marker pulse (Io) ["P600(8), P600(7), P600(6)" = 1] per revolution.

* The home switch will be mounted in such a way that the marker pulse “Io” is always found in the area corresponding to the second home searching feedrate (set by P810, P811, P812).



If there is no room for that, the first home searching feedrate (set by P112, P212, P312) must be reduced. This might be the case with those rotary encoders where the marker pulses are very close to each other.

7.6 SPINDLE

Depending on the setting of machine parameters P601(3) and P601(2), the CNC provides one of the following spindle speed outputs:

* Analog voltage (±10V) via pins 36 and 37 of connector I/O1.
* 2-digit BCD coded output via pins 20 thru 27 of connector I/O1.
* 4-digit BCD coded output via pins 20 thru 27 of connector I/O1.

Analog voltage

To use the CNC's analog voltage for the spindle drive, set P601(3) and P601(2) to "0".

The CNC will generate the analog voltage corresponding to the programmed spindle speed within ±10V.

When a unipolar analog voltage is desired (either 0 to +10V or 0 to -10V), machine parameter P610(4) must be set to "1". The sign of this analog voltage will be set by machine parameter P601(4).

When the machine has an automatic spindle range changer, machine parameter P601(1) must be set to "1". Then, whenever a new spindle speed is selected which involves a range change, the CNC will automatically generate the M function associated with the new spindle speed range M41, M42, M43 or M44.

BCD Output

When desiring a BCD coded output for spindle speed control, machine parameters P601(3) and P601(2) must be set as follows:

For 2-digit BCD output	P601(3)=1 and P601(2)=0
For 4-digit BCD output	P601(3)=0 and P601(2)=1

The CNC will issue the code corresponding to the programmed spindle speed at the BCD outputs (pins 20 thru 27 of I/O 1).

It will also activate the "S Strobe" output to indicate to the electrical cabinet that the required auxiliary function must be executed and it will wait for the "M-DONE" signal from the electrical cabinet in order to consider the data transfer has concluded.

When using a 2-digit BCD code, P601(3)=1 and P601(2)=0, The CNC will indicate the selected spindle speed according to the following conversion table:

Programmed S	S BCD	Programmed S	S BCD	Programmed S	S BCD	Programmed S	S BCD
0	S 00	25-27	S 48	200-223	S 66	1600-1799	S 84
1	S 20	28-31	S 49	224-249	S 67	1800-1999	S 85
2	S 26	32-35	S 50	250-279	S 68	2000-2239	S 86
3	S 29	36-39	S 51	280-314	S 69	2240-2499	S 87
4	S 32	40-44	S 52	315-354	S 70	2500-2799	S 88
5	S 34	45-49	S 53	355-399	S 71	2800-3149	S 89
6	S 35	50-55	S 54	400-449	S 72	3150-3549	S 90
7	S 36	56-62	S 55	450-499	S 73	3550-3999	S 91
8	S 38	63-70	S 56	500-559	S 74	4000-4499	S 92
9	S 39	71-79	S 57	560-629	S 75	4500-4999	S 93
10-11	S 40	80-89	S 58	630-709	S 76	5000-5599	S 94
12	S 41	90-99	S 59	710-799	S 77	5600-6299	S 95
13	S 42	100-111	S 60	800-899	S 78	6300-7099	S 96
14-15	S 43	112-124	S 61	900-999	S 79	7100-7999	S 97
16-17	S 44	125-139	S 62	1000-1119	S 80	8000-8999	S 98
18-19	S 45	140-159	S 63	1120-1249	S 81	9000-9999	S 99
20-22	S 46	160-179	S 64	1250-1399	S 82		
23-24	S 47	180-199	S 65	1400-1599	S 83		

When a value greater than 9999 is programmed, the CNC will indicate the spindle speed code corresponding to 9999.

Example:

When selecting a value of S800, the CNC will issue the BCD code for S78:

	MST80	MST40	MST20	MST10	MST08	MST04	MST02	MST01
Pin	20	21	22	23	24	25	26	27
Value	0	1	1	1	1	0	0	0

When using a 4-digit BCD code, P601(3)=0 and P601(2)=1, the CNC will issue the code corresponding to the programmed S speed in two stages with a 100msec. delay between them.

It will also activate the “S STROBE” signal on each stage and it will wait for the “M-DONE” signal from the electrical cabinet at each stage.

The first stage will issue the values corresponding to the Thousands and Hundreds, and the second stage the ones corresponding to the Tens and Units. The pins corresponding to each one of them are the following:

Pin	1st stage	2nd stage
20 21 22 23	Thousands	Tens
24 25 26 27	Hundreds	Units

Example:

When selecting a value of S 1234, the CNC will show:

PIN	2 Digits (value S81)	4 digits 1st stage	4 digits 2nd stage
20 (MST80)	1	0	0
21 (MST40)	0	0	0
22 (MST20)	0	0	1
23 (MST10)	0	1	1
24 (MST08)	0	0	0
25 (MST04)	0	0	1
26 (MST02)	0	1	0
27 (MST01)	1	0	0

7.6.1 SPINDLE SPEED RANGE CHANGE

With this CNC, the machine can have a gear box in order to adapt the speeds and torques of the spindle motor to the various machining requirements.

Up to 4 spindle ranges may be set by means of machine parameters P7, P8, P9 and P10 specifying the maximum spindle rpm value for each one of them.

The value assigned to P7 must correspond to the lowest range (RANGE 1) and the one assigned to P10 to the highest range (RANGE 4).

When not using all 4 ranges, start the speed assignment from the lowest range up and set the unused ranges to the highest speed being used.

When the new spindle speed selected requires a range change, the CNC will execute the auxiliary M function corresponding to the new range.

The CNC uses the auxiliary functions: M41, M42, M43 and M44 to indicate to the electrical cabinet which range must be selected: RANGE 1, RANGE 2, RANGE 3 or RANGE 4).

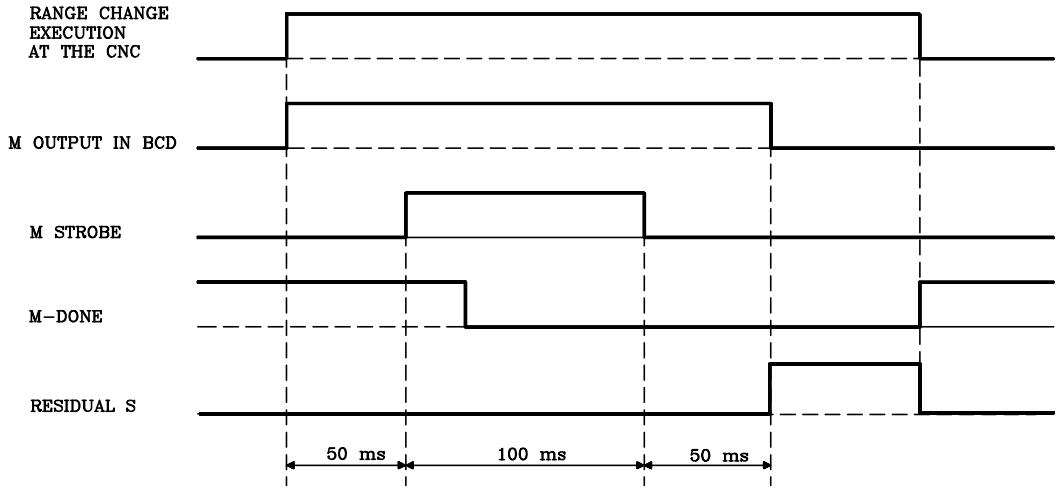
Also, in order to facilitate the range change, the CNC offers the possibility to use a residual analog output during a range change. Machine parameter for the spindle: P601(6).

The value of this residual analog voltage is defined by machine parameter P706 and the oscillation period for this residual analog voltage is set by machine parameter P707.

The automatic range change is carried out as follows:

1.- Once the range change is detected, the CNC outputs the BCD value of the corresponding M function: M41, M42, M43 or M44, via pins 20 thru 27 of connector I/O 1.

50 milliseconds later, it activates the “M Strobe” output to indicate to the electrical cabinet to execute the required M function. This signal is maintained active for 100 milliseconds.



2.- When the electrical cabinet detects the “M Strobe” signal, must deactivate the “M-DONE” input of the CNC to “tell” it that the execution of the corresponding M function has begun.

3.- The electrical cabinet will execute the required M function using the BCD outputs of the CNC (pins 20 thru 27 of connector I/O 1).

4.- After keeping the BCD outputs active for 200 milliseconds, the CNC will output the residual analog output indicated by parameter P706 if so established by parameter P601(6).

The oscillation period for this residual analog voltage is determined by machine parameter P707.

5.- Once the range change is completed, the electrical cabinet must activate the M-DONE input of the CNC to “tell” it that the requested M function has been executed.

Atention:

When the electrical cabinet has some device needing the BCD and “M Strobe” signals from the CNC active for a longer period of time, machine parameter P605(5) must be set “1” (the CNC waits for the down flank of the M-Done signal).

7.7 FEED-HOLD, TRANSFER INHIBIT AND M-DONE SIGNAL PROCESSING

The CNC has one single input (pin 15 of connector I/O 1) to process both signals.

This input must be normally high and these signals are processed as follows:

FEED HOLD

This signal may interrupt the execution of a block.

If while the axes are moving, this input is set low, the CNC maintains the spindle rotating and stops the axes by bringing their analog voltages to "0V" and keeping their enable signals on.

When this input is set back high, the CNC resumes the movement of the axes.

TRANSFER INHIBIT

If this signal is set low while executing a motion block, the CNC interrupts the execution of the program once the current block is executed.

When this signal returns high, the CNC resumes the execution of the program.

M-DONE or CONFIRMATION FROM ELECTRICAL CABINET

This signal is used as confirmation from the electrical cabinet that the execution of the requested M, S or T function has been completed.

When the CNC sends to the electrical cabinet the BCD output signals corresponding to the M, S or T function, the electrical cabinet must set this M-DONE input low.

The CNC will wait for the electrical cabinet to finish the execution of such function and set this M-DONE input back high. This will "tell" the CNC that the execution of the corresponding auxiliary function has been completed.

7.8 AUXILIARY FUNCTIONS M, S, T

M function

This CNC offers up to 100 M functions (M00 thru M99).

The CNC sends out to the electrical cabinet the number of the executed M function via pins 20 thru 27 of connector I/O1. Machine parameter P617(8) determines whether this value is sent out in BCD or binary code.

Also, it must be borne in mind that the CNC has an internal "decoded M" table. The way to operate with this table is described later in this chapter.

Every time an M function of the decoded-M table is executed, the CNC updates its corresponding outputs at connector I/O2.

Machine parameter P609(5) determines whether the CNC also outputs their corresponding BCD or binary coded number via pins 20 thru 27 of connector I/O1.

S function

Only to be used when the spindle speed output is in BCD (not analog). Machine parameter P601(3)=1.

Whenever a block containing a new spindle speed "S" is executed, the CNC will output the corresponding BCD code via pins 20 thru 27 of connector I/O 1.

T function

Whenever a block containing a new tool "T" is executed, the CNC will output the corresponding BCD code via pins 20 thru 27 of connector I/O 1.

This CNC also offers a tool table where the length and radius of each tool can be defined. The CNC will take these dimensions into account when machining with tool compensation (G41, G42, G43).

The way to operate with this table is described in the chapter on "Auxiliary Functions" in this manual.

7.8.1 DECODED M FUNCTION TABLE

To access this table, press the following keystroke sequence:

[AUX]	(SPECIAL FUNCTIONS)
[5]	(AUXILIARY MODES)
[1]	(SPECIAL MODES)
[0] [1] [0] [1]	(Access code)
[2]	(AUXILIARY M FUNCTIONS)

The CNC shows the following information for each of the M functions set in this table:

M41	100100100100100	(outputs to be activated)
	00100100100100100	(outputs to be deactivated)

* The number of the M function already set. Mxx indicates that this position is free and any other M function may be set.

* The first row has 15 characters. Each one corresponds to a decoded-M output of connector I/O2 and their value (0 or 1) indicates the following:

0 This M function must **not activate** its decoded output at connector I/O2.
1 This M function **must activate** its decoded output at connector I/O2.

* The second row has 17 characters. The first 15 (from left to right) correspond to the decoded-M output of I/O2 and their value (0 or 1) indicates the following:

0 This M function must **not deactivate** its decoded output at connector I/O2.
1 This M function **must deactivate** its decoded output at connector I/O2.

Bit 16 determines whether the M function is executed at the beginning (if 0) or at the end of the block (if 1) where it is programmed. If this bit is set to "1" and the programmed block contains a movement of axes, the M function will be executed once the axes have reached their target position.

Bit 17 determines the way the M function is transferred to the electrical cabinet. See the following section for its description.

For example: If the table corresponding to function M41 has been set as follows:

M41	100100100100100	(outputs to be activated)
	00100100100100100	(outputs to be deactivated)

The CNC will behave as follows whenever M41 is executed:

	M01	M02	M03	M04	M05	M06	M07	M08	M09	M10	M11	M12	M13	M14	M15
Pin I/O2	3	4	5	6	7	8	9	10	11	12	13	25	24	23	22
at 24V	x			x			x			x			x		
at 0V			x			x			x			x			x
Not modified		x			x			x			x			x	

7.8.2 M, S, T FUNCTION TRANSFER

Every time a block is executed, the CNC transfers to the electrical cabinet information of the M, S, T functions activated in it.

First, the M function information is transferred, then, that of the S functions and, then, that of the T functions.

M function:

The CNC analyzes the M functions programmed in the block and, depending on how they are defined, they will be output before or after moving the axes.

When sending more than one M function, the CNC will send them one by one starting from the first one in the block and, once it has been sent out, it will output the next one and so forth.

When sending the auxiliary M functions to the electrical cabinet, the CNC uses the BCD outputs (pins 20 thru 27 of connector I/O 1) and it activates the “M Strobe” output to “tell” the electrical cabinet to execute them.

Depending on how they are defined on the table, the CNC must wait or not for the “M-DONE” signal to consider it executed.

If an M function which is not defined on the M function table is executed, the CNC will wait for the “M-DONE” signal before resuming the execution of the program.

S function transfer:

When a new spindle speed (S) is programmed, the CNC issues the corresponding BCD code (via pins 20 thru 27 of connector I/O 1) and it activates the “S Strobe” output to “tell” the electrical cabinet to execute it.

The CNC will wait for the “M-DONE” signal to consider it executed.

If the new selected “S” involves a range change, the CNC executes first the M functions corresponding to the range change and then, it transfers the new selected spindle speed.

T function transfer:

When selecting a new tool “T”, the CNC issues the corresponding BCD code (via pins 20 thru 27 of I/O 1) and it will activate the “T Strobe” to “tell” the electrical cabinet to execute it.

The CNC will wait for the “M-DONE” signal to consider it executed.

7.8.3 M, S, T FUNCTION TRANSFER USING THE M-DONE SIGNAL

When parameter P605(5) is set to “0”, the CNC maintains the BCD outputs and the corresponding Strobe signal (M, S, T) active for 100 milliseconds.

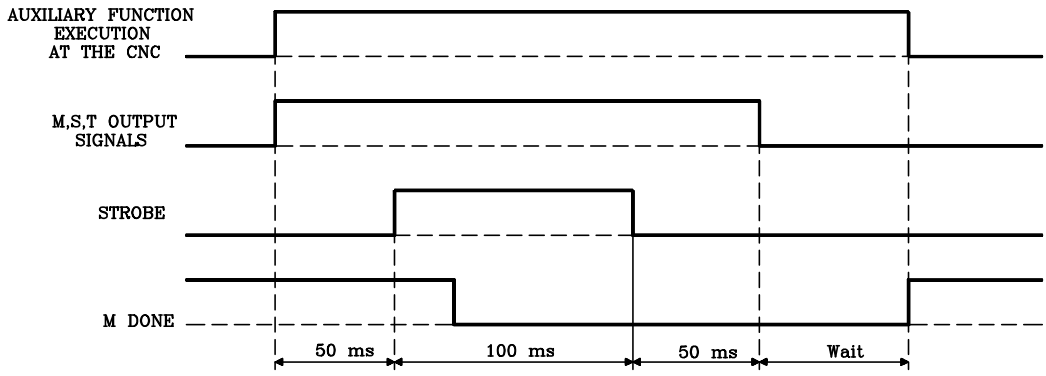
When the electrical cabinet has a device requiring the BCD signals to be active for a longer time, machine parameter P605(5) must be set to “1”.

In each case, the CNC acts as follows:

“P605(5)=0”

1.- The CNC transfers the BCD value of the selected function via pins 20 thru 27 of connector I/O 1.

50 milliseconds later, the “M Strobe” output is activated to “tell” the electrical cabinet to execute the M function.



2.- When the electrical cabinet detects the activation of the “M Strobe” signal, it must start the execution of the corresponding function.

3.- The CNC will maintain the “M Strobe” signal for 100 milliseconds and the BCD signals for another 50 milliseconds.

After this time period, it will wait for the M-DONE signal provided by the electrical cabinet indicating to the CNC that the execution of the “M” function is completed.

If the M-done signal was not deactivated in point 2, the CNC will consider the M transfer completed after the BCD signals disappear (there is no wait).

Atention:

When executing an M function which has been set at the decoded-M table, the CNC analyzes the status of bit 17 of the bottom row.

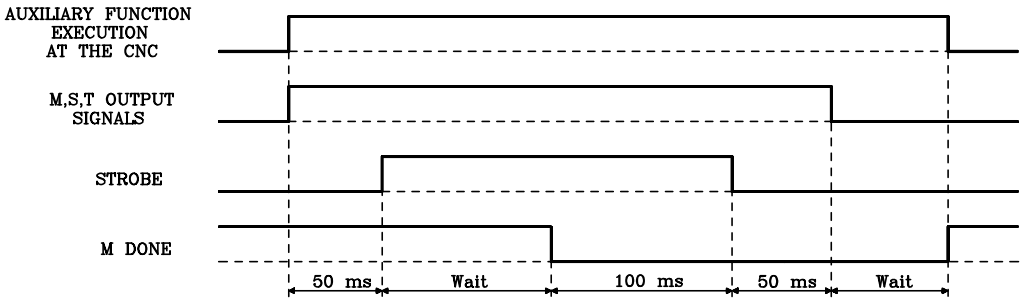
If "0", it behaves as described above; but if "1", it behaves as described next [P605(5)=1]

“P605(5)=1”

This type of transfer is used when the electrical cabinet has a device which requires the BCD outputs from the CNC to be active for a longer period of time.

1.- The CNC sends the BCD value of the selected function via pins 20 thru 27 of connector I/O 1.

50 milliseconds later, it activates the corresponding Strobe output to “tell” the electrical cabinet to execute the required auxiliary function.



2.- When the electrical cabinet detects the activation of one of the Strobe signals, it must begin the execution of the corresponding function deactivating the M-DONE signal to let the CNC know that this M function execution has begun.

3.- The CNC will maintain the Strobe signal for another 100 milliseconds and the BCD outputs for another 150 milliseconds.

After this time period, it will wait for the electrical cabinet to reactivate the M-DONE input “telling” the CNC that the processing of the required function has concluded.

Atention:

When machine parameter P605(5) = 1 and an M function is executed which is set in the decoded-M table, the CNC ignores the status of bit 17 of the bottom row.

APPENDIX A

CNC TECHNICAL CHARACTERISTICS

GENERAL CHARACTERISTICS

Three 8-bit microprocessors
Memory capacity for up to 7 part-programs of 20 operations each.
1 communication line: RS232C.
6 feedback inputs up to 3 axes + 3 electronic handwheels.
Resolution of 0.001 mm. or 0.0001 inches.
Multiplying factor of up to x100 for sine-wave feedback signals.
Feedrates from 0.001 mm/min up to 65535 mm/min (0.0001 through 2580 inches/min)
Maximum axis travel ±8388.607 mm (330.2601 inches)
11 optocoupled digital inputs.
32 optocoupled digital outputs.
4 analog outputs: ±10V (one per axis + spindle).
Approximate weight: 12Kg.
Maximum consumption in normal operation: 75W

PACKAGING

Meets the "EN 60068-2-32" standard.

POWER SUPPLY

High performance Switching power supply.
Universal power supply with any input between 100 V AC and 240 V AC (±10% and -15%).
AC frequency: 50 - 60 Hz ±1% and ±2% during very short periods.
Power outages. Meets the EN 61000-4-11 standard. It is capable of withstanding micro outages of up to 10 milliseconds.
Harmonic distortion: Less than 10% of the rms voltage between low voltage conductors (sum of the 2^{nd} through the 5^{th} harmonic)

ELECTRICAL CHARACTERISTICS OF FEEDBACK INPUTS

+5V power consumption: 750 mA (250 mA per axis).
-5V power consumption: 0.3A (100 mA per axis).
Operating levels for square-wave signals:
- Maximum frequency: 200KHz.
- Maximum separation between flanks: 950 nsec.
- Phase shift: 90° ±20°
- High threshold (logic state "1"): 2.4V. < V_{IH} < 5V.
- Low threshold (logic state "0"): -5V. < V_{IL} < 0.8V.
- Vmax.: ±7 V.
- Hystheresis: 0.25 V.
- Maximum input current: 3mA.

Operating levels for sine-wave signals:
- Maximum frequency: 25KHz.
- Peak to peak voltage: 2V. < Vpp < 6V.
- Input current I_I 1mA.

ELECTRICAL CHARACTERISTICS OF DIGITAL INPUTS

Nominal voltage: +24 V DC. (Maximum: +30Vdc; Minimum: +18Vdc)
High threshold (logic state "1"): V_{IH} >+18 V DC.
Low threshold (logic state "0"): V_{IL} < +5 V DC. or not connected.
Typical consumption per input: 5 mA. (Maximum: 7 mA)
Protection by means of galvanic isolation by opto-couplers.
Protection against reversed connection up to -30 V DC.

ELECTRICAL CHARACTERISTICS OF DIGITAL OUTPUTS

Nominal voltage: +24 V DC. (Maximum: +30Vdc; Minimum: +18Vdc)
Output voltage Vout = Power Supply voltage - 2 V DC.
Maximum output current: 100 mA.
Protection by means of galvanic isolation by opto-couplers.
Protection by means of external 3Amp fuse against reversed connection up to -30 V DC and overvoltage of the external power supply greater than 33V DC.

CRT

Monitor	8" monochrome	Deflection:	90 degrees
Screen:	Anti-glare	Phosphor:	PLA (amber)
Resolution:	600 lines	Display surface:	146 x 119 mm.

SWEEP FREQUENCY

Vertical synchronism: 50-60 Hz positive Horizontal synchronism: 19.2 KHz positive

AMBIENT CONDITIONS

Relative humidity: 30-95% non condensing
Operating temperature: 5ºC - 40ºC (41º F - 104ºF) with an average lower than 35ºC (95º F)
Storage temperature : between 25º C (77ºF and 70º C (158º F).
Maximum operating altitude : Meets the "IEC 1131-2" standard.

VIBRATION

Under working conditions: 10-50 Hz. amplitude 0.2 mm.
Under transport conditions: 10-50 Hz. amplitude 1 mm, 50-300 Hz. and acceleration of 5g.
Free fall of packaged equipment: 1 m.

ELECTROMAGNETIC COMPATIBILITY

See Declaration of Conformity in the introduction of this manual.

SAFETY

See Declaration of Conformity in the introduction of this manual

DEGREE OF PROTECTION

Central Unit : IP 54
Accessible parts inside the enclosure: IP 1X

The machine manufacturer must comply with the "EN 60204-1 (IEC-204-1)", standard regarding protection against electrical shock due to I/O contact failures with external power supply when not hooking up this connector before turning the power supply on.
Access to the inside of the unit is absolutely forbidden to non authorized personnel.

BATTERY

3.5V lithium battery.

Estimated life: 10 years

As from error indication (low battery), the information contained in memory will be kept for a maximum of 10 days with the CNC off. It must be replaced.

Caution, due to risk of explosion or combustion:

Do not attempt to recharge the battery.
Do not expose it to temperatures over 100 °C (232°F).
Do not short-circuit its leads.

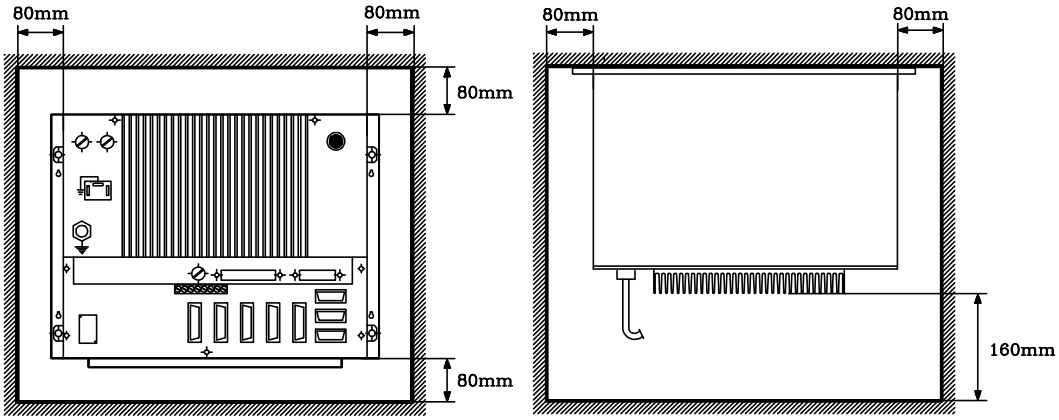
Atention:

To avoid excessive heating of internal circuits, the several ventilation slits must not be obstructed. It is also necessary to install a ventilation system which extracts hot air from the enclosure or desk supporting the CNC.

APPENDIX B

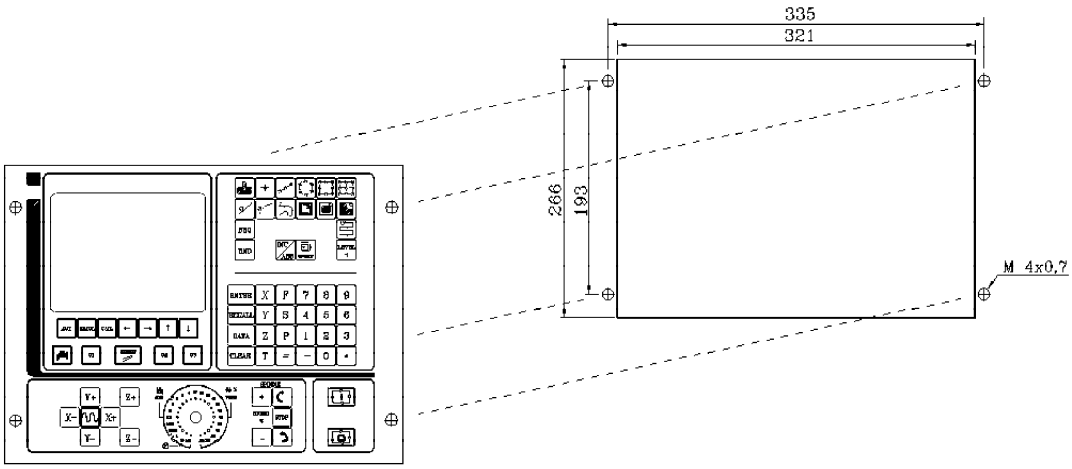
ENCLOSURES

The minimum distance between the sides of the CNC and its enclosure in order to meet the required ambient conditions must be the following:



When using a fan to better ventilate the enclosure, a **DC FAN** must be used since an AC fan may generate electromagnetic interference resulting in distorted images being displayed by the CRT.

The CNC must be secured as shown below (dimensions in mm):



APPENDIX C

CNC INPUTS AND OUTPUTS

INPUTS

Pin	Connector	Function
10	I/O 1	X axis Home switch
11	I/O 1	Y axis Home switch
12	I/O 1	Z axis Home switch
14	I/O 1	Emergency stop.
15	I/O 1	/Feed hold - /Transfer inhibit - /M-done
16	I/O 1	/Stop
17	I/O 1	Start - rapid jog
18	I/O 1	Conditional input (block skip)
19	I/O 1	DRO mode

OUTPUTS

Pin	Connector	Function
2	I/O 1	T Strobe
3	I/O 1	S Strobe
4	I/O 1	M Strobe
5	I/O 1	Emergency
6	I/O 1	Threading on
7	I/O 1	Z axis enable
8	I/O 1	Y axis enable
9	I/O 1	X axis enable
20	I/O 1	MST80
21	I/O 1	MST40
22	I/O 1	MST20
23	I/O 1	MST10
24	I/O 1	MST08
25	I/O 1	MST04
26	I/O 1	MST02
27	I/O 1	MST01
30, 31	I/O 1	X axis analog output
32, 33	I/O 1	Y axis analog output
34, 35	I/O 1	Z axis analog output
36, 37	I/O 1	Spindle analog output
3	I/O 2	Decoded M01 output. Coolant
4	I/O 2	Decoded M02 output
5	I/O 2	Decoded M03 output
6	I/O 2	Decoded M04 output
7	I/O 2	Decoded M05 output
8	I/O 2	Decoded M06 output
9	I/O 2	Decoded M07 output
10	I/O 2	Decoded M08 output
11	I/O 2	Decoded M09 output
12	I/O 2	Decoded M10 output
13	I/O 2	Decoded M11 output
21	I/O 2	"Jog mode selected" output
22	I/O 2	Decoded M15 output
23	I/O 2	Decoded M14 output - Reset
24	I/O 2	Decoded M13 output - Cycle on - Automatic - G00
25	I/O 2	Decoded M12 output - Vertical axis movement

APPENDIX D

2-DIGIT BCD CODED "S" OUTPUT CONVERSION TABLE

Programmed S	S BCD	Programmed S	S BCD	Programmed S	S BCD	Programmed S	S BCD
0	S 00	25-27	S 48	200-223	S 66	1600-1799	S 84
1	S 20	28-31	S 49	224-249	S 67	1800-1999	S 85
2	S 26	32-35	S 50	250-279	S 68	2000-2239	S 86
3	S 29	36-39	S 51	280-314	S 69	2240-2499	S 87
4	S 32	40-44	S 52	315-354	S 70	2500-2799	S 88
5	S 34	45-49	S 53	355-399	S 71	2800-3149	S 89
6	S 35	50-55	S 54	400-449	S 72	3150-3549	S 90
7	S 36	56-62	S 55	450-499	S 73	3550-3999	S 91
8	S 38	63-70	S 56	500-559	S 74	4000-4499	S 92
9	S 39	71-79	S 57	560-629	S 75	4500-4999	S 93
10-11	S 40	80-89	S 58	630-709	S 76	5000-5599	S 94
12	S 41	90-99	S 59	710-799	S 77	5600-6299	S 95
13	S 42	100-111	S 60	800-899	S 78	6300-7099	S 96
14-15	S 43	112-124	S 61	900-999	S 79	7100-7999	S 97
16-17	S 44	125-139	S 62	1000-1119	S 80	8000-8999	S 98
18-19	S 45	140-159	S 63	1120-1249	S 81	9000-9999	S 99
20-22	S 46	160-179	S 64	1250-1399	S 82		
23-24	S 47	180-199	S 65	1400-1599	S 83		

APPENDIX E

MACHINE PARAMETER SUMMARY CHART

GENERAL MACHINE PARAMETERS

P5	Mains frequency (50/60)	*Section 4.3*
P99	Language (0=Spanish, 1=German, 2=English, 3=French, 4=Italian)	
P13	Measuring units: mm (0), inches (1)	
P6	Theoretical (1) or Real (0) display	
P617(5), P605(6), P617(4)	X, Y, Z as DRO axis (0=No, 1=Yes)	
P618(6), P618(5), P618(4)	X, Y, Z axis display (1=No, 0=Yes)	
P701	Number of tools (0..98)	
P626(1)	The CNC displays the tool base (1) or tool tip (0) position	
P743	Subroutine to be executed before the T function	
P745	Subroutine to be executed after the "T" function.	
P628(1)	Display of the following error (0=No, 1=Yes)	
P628(6)	O1 may be turned on/off only when spindle is off (0=No, 1=Yes)	

I/O RELATED MACHINE PARAMETERS *Section 4.3.1*

P605(8)	Normal status of Emergency output (pin 5 of I/O1). (0=0V, 1=24V)
P605(5)	The CNC waits for trailing edge of "M-done" input (0=No, 1=Yes)
P609(7)	Pin 17 of I/O 1 as "Rapid jog" (0=No, 1=Yes)
P609(3)	Pin 23 of I/O 2 as "RESET" (0=No, 1=Yes)
P611(1)	Pin 24 of I/O 2 as "Program block in execution" (0=No, 1=Yes)
P611(6)	Pin 24 of I/O 2 as "P99996 in execution" (0=No, 1=Yes)
P613(4)	Pin 24 of I/O 2 as "Rapid traverse" (0=No, 1=Yes)
P613(2)	Pin 25 of I/O 2 as "Vertical move" output (0=No, 1=Yes)
P617(8)	M function output in BCD (0) or Binary (1)
P609(5)	M functions set on the "M" table are output in BCD (0) or binary (1)
P602(8,7,6,5), P603(1)	Feedback alarm cancellation connect.: A1, A2, A3, A4, A5. (0=No,1=Yes)

HANDWHEEL MACHINE PARAMETERS *Section 4.3.2*

P612(1), P626(8), P627(8)	Electronic handwheel associated with X, Y, Z (0=No, 1=Yes)
P613(1)	Electronic handwheel model FAGOR 100P (0=No, 1=Yes)
P612(2), P626(2), P627(2)	Counting direction of the electronic handwheel for X, Y, Z
P612(3), P626(3), P627(3)	Feedback units of electronic handwheels X,Y,Z (0=mm, 1=inch)
P612(4,5), P626(4,5), P627(4,5)	Counting resolution of the electronic handwheel for X, Y, Z
P612(6), P626(6), P627(6)	Multiplying factor for handwheel signals X,Y,Z (0=x4, 1=x2)
P628(5)	Handwheels only operate at handwheel positions of FOS (1=No, 0=Yes)
P628(8)	The machine uses mechanical handwheels (0=No, 1=Yes)
P815	Delay before opening the closed loop. (1=10ms)

MACHINE PARAMETERS RELATED TO OPERATING MODE *Section 4.3.3*

P609(8)	Axis orientation in graphic representation (0=Mill, 1=Boring mill)
P605(4)	Axis orientation in XZ plane
P628(7)	Z interpolation (1=No, 0=Yes)
P618(1)	CYCLE START key inhibit (0=No, 1=Yes)
P606(3)	M30 executed when switching to Jog mode (0=No, 1=Yes)
P12	Continuous (0) or pulsating (1) axis jog
P609(6)	Maximum incremental Jog (0=10mm or 1 inch, 1=1mm or 0.1 inch)
P606(2)	Maximum Feedrate Override Switch value applied by the CNC (0=120%, 1=100%)
P4	Feedrate Override Switch active in G00 (0=No, 1=Yes)
P613(5)	G05 or G07 on CNC power-up (0=G07, 1=G05)
P715	Dwell between G07 blocks (square corner) (1=10ms)
P611(5)	Feedrate units in G94 (0=1mm/min or 0.1 inch/min, 1=0.1mm/min or 0.01 inch/min)
P610(1)	Feed-Hold in G47 (0=No, 1=Yes)
P628(4)	The "rapid jog" key applies a feedrate override range over 100% (0=No, 1=Yes)

MACHINE PARAMETERS RELATED WITH RS232C SERIAL LINE *Section 4.3.4*

P0	Communications speed (baudrate) (110, 150, 300, 600, 1200, 2400, 4800, 9600)
P1	Number of data bits per character (7/8)
P2	Parity (0= No, 1= Odd, 2= Even)
P3	Stop bits (1/2)
P607(3)	DNC (0=No, 1=Yes)
P607(4)	Communications settings. FAGOR cassette (0) or FAGOR Floppy Disk Unit (1)
P607(5)	DNC protocol active on power-up (0=No, 1=Yes)
P607(6)	The CNC aborts DNC communications (program debugging) (1=No, 0=Yes)
P607(7)	Status report by interruption (0=No, 1=Yes)

MACHINE PARAMETERS FOR THE AXES

P100, P200, P300	Sign of the analog output for X, Y, Z	*Section 5.*
P101, P201, P301	Counting direction for X, Y, Z	
P102, P202, P302	Jogging direction for X, Y, Z	

MACHINE PARAMETERS FOR AXIS RESOLUTION *Section 5.1*

P103, P203, P303	Counting resolution for X, Y, Z
P622(1), P622(2), P622(3)	Resolution units for X, Y, Z sine-wave feedback
P604(4), P604(3), P604(2)	Feedback units for X, Y, Z (0=mm, 1=inch)
P106, P206, P306	Feedback signal type for X, Y, Z (0=Squarewave, 1=Sinewave)
P604(8), P604(7), P604(6)	Multiplying factor for X, Y, Z feedback signals (0=x4, 1=x2)
P603(8), P603(7), P603(6)	Binary encoder for X, Y, Z (0=No, 1=Yes)
P610(8), P610(7), P610(6)	Equivalence of the binary encoder for X, Y, Z

MACHINE PARAMETERS FOR ANALOG OUTPUTS *Section 5.2*

P117, P217, P317	Minimum analog output for X, Y, Z (1=2.5mV)
P104, P204, P304	Dwell between Enable and Analog output for X, Y, Z (0=No, 1=Yes)
P118, P218, P318	In-position zone (dead-band) for X, Y, Z (0..255 microns)
P105, P205, P305	Continuous control of X, Y, Z (0=No, 1=Yes)

MACHINE PARAMETERS FOR AXIS TRAVEL LIMITS *Section 5.3*

P107, P207, P307	Positive travel limit for X, Y, Z
P108, P208, P308	Negative travel limit for X, Y, Z

FEEDRATE RELATED MACHINE PARAMETERS *Section 5.4*

P110, P210, P310	Maximum programmable feedrate X, Y, Z
P111, P211, P311	G00 feedrate for X, Y, Z
P729	Maximum feedrate for circular interpolations
P708	Feedrate/Override when analog output reaches 10V.
P714	Error if actual feedrate not within 50% to 200% of programmed value
P615(6)	Programmed feedrate in mm/min (0) or inches/minute (1)

MACHINE PARAMETERS FOR AXIS CONTROL *Section 5.5*

P114, P214, P314	Proportional gain K1 for X, Y, Z
P115, P215, P315	Gain break point for X, Y, Z
P116, P216, P316	Proportional gain K2 for X, Y, Z
P611(8)	In G00 and F00, proportional gain K2 from 256 microns on (0=No, 1=Yes)
P726	Recovery of programmed position on "non-continuously" controlled axes

HOME SEARCH RELATED MACHINE PARAMETERS *Section 5.6*

P119, P219, P319	Home coordinates for X, Y, Z
P623(8), P623(7), P623(6)	Home search direction for X, Y, Z (0=Positive, 1=Negative)
P600(8), P600(7), P600(6)	Home pulse (marker) type for X, Y, Z (1=Positive, 0=Negative)
P602(4), P602(3), P602(2)	Home switch for X, Y, Z (1=No, 0=Yes)
P112, P212, P312	1st homing feedrate for X, Y, Z
P810, P811, P812	2nd homing feedrate for X, Y, Z
P611(2)	Mandatory home search on power-up (0=No, 1=Yes)
P804	Jogging feedrate prior to mandatory home search
P606(4)	Home search function generates an M30 (0=No, 1=Yes)

ACCELERATION/DECELERATION RELATED MACHINE PARAMETERS *Section 5.7*

P721, P722, P723	Apply ACC/DEC onto X, Y, Z (1=20ms)
P613(7)	Linear ACC/DEC on all linear interpolation (G01) (0=No, 1=Yes)
P620(2)	Linear ACC/DEC in G05 (round corner) (1=No, 0=Yes)
P624(8)	Bell-shaped ACC/DEC (0=No, 1=Yes)
P744	Bell-shaped ACC/DEC ramp duration (1=10ms)
P732, P733, P734	FEED-FORWARD gain for X, Y, Z

LEADSCREW RELATED MACHINE PARAMETERS *Section 5.8*

P109, P209, P309	Leadscrew backlash compensation for X, Y, Z (0..255 microns)
P624(1), P624(2), P624(3)	Sign of leadscrew backlash compensation X, Y, Z (0=Pos., 1=Neg.)
P113, P213, P313	Additional analog pulse for X, Y, Z (1=2.5mV)
P606(8), P606(7), P606(6)	Leadscrew error compensation for X, Y, Z (0=No, 1=Yes)

SPECIAL MACHINE PARAMETERS *Section 5.9*

P609(1)	Machine with travels over 8 meters (0=No, 1=Yes)
P617(6)	Resolution of 0.0001 millimeters (0.00001 inches). (0=No, 1=Yes)
P908, P909	Collision zone for Y and Z

SPINDLE MACHINE PARAMETERS

MACHINE PARAMETERS FOR SPINDLE RANGE CHANGE *Section 6.1*

P7, P8, P9, P10	Maximum speed for ranges 1, 2, 3 and 4 (0..9999 rpm)
P601(6)	Residual analog voltage "S" for range change (0=No, 1=Yes)
P706	Value of the residual analog voltage (1=2.5mV)
P707	Oscillation period during range change

MACHINE PARAMETERS FOR ANALOG SPINDLE SPEED OUTPUT *Section 6.2*

P601(4)	Sign of the analog spindle speed output "S"
P610(4)	Unipolar (1) or bipolar (0) analog spindle speed output "S"
P609(4)	All spindle **speed** changes generate an "S Strobe" signal (0=No, 1=Yes)

MACHINE PARAMETERS FOR BCD-CODED SPINDLE SPEED OUTPUT *Section 6.3*

P601(3)	2-digit BCD coded output (0=No, 1=Yes)
P601(2)	4-digit BCD coded output (0=No, 1=Yes)

APPENDIX F

SEQUENTIAL MACHINE PARAMETER LIST

P0	Communications baudrate (110, 150, 300, 600, 2400, 4800, 9600)	*Section 4.3.4*
P1	Communications data bits (7/8)	*Section 4.3.4*
P2	Parity (0=No, 1=Odd, 2=Even)	*Section 4.3.4*
P3	Stop bits (1/2)	*Section 4.3.4*
P4	Feedrate Override active in G00 (0=No, 1=Yes)	*Section 4.3.3*
P5	Mains (AC) frequency (50/60)	*Section 4.3*
P6	Theoretical (1) or real (0) display	*Section 4.3*
P7	Maximum spindle speed for RANGE 1 (0..9999 rpm)	*Section 6.1*
P8	Maximum spindle speed for RANGE 2 (0..9999 rpm)	*Section 6.1*
P9	Maximum spindle speed for RANGE 3 (0..9999 rpm)	*Section 6.1*
P10	Maximum spindle speed for RANGE 4 (0..9999 rpm)	*Section 6.1*
P11	*Not being used at this time "= 0"*	
P12	Continuous (0) or pulsating (1) axis JOG	*Section 4.3.3*
P13	Measuring units. (0= mm, 1= inches)	*Section 4.3*
P99	Language (0=Spanish, 1= German, 2=English, 3=French, 4=Italian)	*Section 4.3*
P100	Sign of the X axis analog output	*Section 5.*
P101	Counting direction of the X axis	*Section 5.*
P102	X axis jogging direction	*Section 5.*
P103	X axis feedback (counting) resolution	*Section 5.1*
P104	Dwell between Enable and analog output for the X axis (0=No, 1=Yes)	*Section 5.2*
P105	Continuous control of the X axis (0=No, 1=Yes)	*Section 5.2*
P106	Type of feedback signal for the X axis (0=Squarewave, 1=Sinewave)	*Section 5.1*
P107	X axis positive travel limit	*Section 5.3*
P108	X axis negative travel limit	*Section 5.3*
P109	X axis leadscrew backlash (0..255 microns)	*Section 5.8*
P110	X axis maximum programmable feedrate	*Section 5.4*
P111	X axis G00 feedrate	*Section 5.4*
P112	1st home searching feedrate for the X axis	*Section 5.6*
P113	Additional analog pulse for the X axis (1=2.5mV)	*Section 5.8*
P114	Proportional gain K1 for the X axis	*Section 5.5*
P115	Gain break point for the X axis	*Section 5.5*
P116	Proportional gain K2 for the X axis	*Section 5.5*
P117	Minimum X axis analog (1=2.5mV)	*Section 5.2*
P118	In-position zone (dead band) for the X axis (0..255 microns)	*Section 5.2*
P119	X axis home coordinate	*Section 5.6*
P200	Sign of the Y axis analog output	*Section 5.*
P201	Counting direction of the Y axis	*Section 5.*
P202	Y axis jogging direction	*Section 5.*
P203	Y axis feedback (counting) resolution	*Section 5.1*
P204	Dwell between Enable and analog output for the Y axis (0=No, 1=Yes)	*Section 5.2*
P205	Continuous control of the Y axis (0=No, 1=Yes)	*Section 5.2*
P206	Type of feedback signal for the Y axis (0=Squarewave, 1=Sinewave)	*Section 5.1*
P207	Y axis positive travel limit	*Section 5.3*
P208	Y axis negative travel limit	*Section 5.3*
P209	Y axis leadscrew backlash (0..255 microns)	*Section 5.8*
P210	Y axis maximum programmable feedrate	*Section 5.4*
P211	Y axis G00 feedrate	*Section 5.4*
P212	1st home searching feedrate for the Y axis	*Section 5.6*
P213	Additional analog pulse for the Y axis (1=2.5mV)	*Section 5.8*
P214	Proportional gain K1 for the Y axis	*Section 5.5*
P215	Gain break point for the Y axis	*Section 5.5*
P216	Proportional gain K2 for the Y axis	*Section 5.5*
P217	Minimum Y axis analog (1=2.5mV)	*Section 5.2*
P218	In-position zone (dead band) for the Y axis (0..255 microns)	*Section 5.2*
P219	Y axis home coordinate	*Section 5.6*
P300	Sign of the Z axis analog output	*Section 5.*
P301	Counting direction of the Z axis	*Section 5.*
P302	Z axis jogging direction	*Section 5.*
P303	Z axis feedback (counting) resolution	*Section 5.1*
P304	Dwell between Enable and analog output for the Z axis (0=No, 1=Yes)	*Section 5.2*
P305	Continuous control of the Z axis (0=No, 1=Yes)	*Section 5.2*
P306	Type of feedback signal for the Z axis (0=Squarewave, 1=Sinewave)	*Section 5.1*
P307	Z axis positive travel limit	*Section 5.3*
P308	Z axis negative travel limit	*Section 5.3*
P309	Z axis leadscrew backlash (0..255 microns)	*Section 5.8*

P310 Z axis maximum programmable feedrate ..... *Section 5.4*
P311 Z axis G00 feedrate ..... *Section 5.4*
P312 1st home searching feedrate for the Z axis ..... *Section 5.6*
P313 Additional analog pulse for the Z axis (1=2.5mV) ..... *Section 5.8*
P314 Proportional gain K1 for the Z axis ..... *Section 5.5*
P315 Gain break point for the Z axis ..... *Section 5.5*
P316 Proportional gain K2 for the Z axis ..... *Section 5.5*
P317 Minimum Z axis analog (1=2.5mV) ..... *Section 5.2*
P318 In-position zone (dead band) for the Z axis (0..255 microns) ..... *Section 5.2*
P319 Z axis home coordinate ..... *Section 5.6*

P400 thru P519 *Not being used at this time "=0"*

P600(8) Type of X axis marker pulse (home) (0=Negative, 1=Positive) ..... *Section 5.6*
(7) Type of Y axis marker pulse (home) (0=Negative, 1=Positive) ..... *Section 5.6*
(6) Type of Z axis marker pulse (home) (0=Negative, 1=Positive) ..... *Section 5.6*
(5) *Not being used at this time "=0"*
(4) *Not being used at this time "=0"*
(3) *Not being used at this time "=0"*
(2) *Not being used at this time "=0"*
(1) *Not being used at this time "=0"*

P601(8) *Not being used at this time "=0"*
(7) *Not being used at this time "=0"*
(6) Residual analog S output during range change (0=No, 1=Yes) ..... *Section 6.1*
(5) *Not being used at this time "=0"*
(4) Sign of the spindle analog output ..... *Section 6.2*
(3) S output in 2-digit BCD code (0=No, 1=Yes) ..... *Section 6.3*
(2) S output in 2-digit BCD code (0=No, 1=Yes) ..... *Section 6.3*
(1) *Not being used at this time "=0"*

P602(8) Cancellation of A1 feedback alarm (0=No, 1=Yes) ..... *Section 4.3.1*
(7) Cancellation of A2 feedback alarm (0=No, 1=Yes) ..... *Section 4.3.1*
(6) Cancellation of A3 feedback alarm (0=No, 1=Yes) ..... *Section 4.3.1*
(5) Cancellation of A4 feedback alarm (0=No, 1=Yes) ..... *Section 4.3.1*
(4) X axis home switch (1=No, 0=Yes) ..... *Section 5.6*
(3) Y axis home switch (1=No, 0=Yes) ..... *Section 5.6*
(2) Z axis home switch (1=No, 0=Yes) ..... *Section 5.6*
(1) *Not being used at this time "=0"*

P603(8) X axis with binary encoder (0=No, 1=Yes) ..... *Section 5.1*
(7) Y axis with binary encoder (0=No, 1=Yes) ..... *Section 5.1*
(6) Z axis with binary encoder (0=No, 1=Yes) ..... *Section 5.1*
(5) *Not being used at this time "=0"*
(4) *Not being used at this time "=0"*
(3) *Not being used at this time "=0"*
(2) *Not being used at this time "=0"*
(1) Cancellation of A5 feedback alarm (0=No, 1=Yes) ..... *Section 4.3.1*

P604(8) X axis feedback signal multiplying factor (0=x4, 1=x2) ..... *Section 5.1*
(7) Y axis feedback signal multiplying factor (0=x4, 1=x2) ..... *Section 5.1*
(6) Z axis feedback signal multiplying factor (0=x4, 1=x2) ..... *Section 5.1*
(5) *Not being used at this time "=0"*
(4) X axis feedback units (0=mm, 1=inch) ..... *Section 5.1*
(3) Y axis feedback units (0=mm, 1=inch) ..... *Section 5.1*
(2) Z axis feedback units (0=mm, 1=inch) ..... *Section 5.1*
(1) *Not being used at this time "=0"*

P605(8) Normal status of Emergency output at pin 5 of I/O1. (0=5V, 1=24V) ..... *Section 4.3.1*
(7) *Not being used at this time "=0"*
(6) Y axis as DRO axis (0=No, 1=Yes) ..... *Section 4.3*
(5) CNC waits for trailing edge at "M-done" input (0=No, 1=Yes) ..... *Section 4.3.1*
(4) Axis orientation on the XZ plane ..... *Section 4.3.3*
(3) *Not being used at this time "=0"*
(2) *Not being used at this time "=0"*
(1) *Not being used at this time "=0"*

P606(8) X axis leadscrew error compensation (0=No, 1=Yes) ..... *Section 5.8*
(7) Y axis leadscrew error compensation (0=No, 1=Yes) ..... *Section 5.8*
(6) Z axis leadscrew error compensation (0=No, 1=Yes) ..... *Section 5.8*
(5) *Not being used at this time "=0"*
(4) Home search function generates an M30 (0=No, 1=Yes) ..... *Section 5.6*
(3) M30 executed when switching to JOG mode (0=No, 1=Yes) ..... *Section 4.3.3*
(2) Maximum Feedrate override value applied by CNC (0=120%, 1=100%) ..... *Section 4.3.3*
(1) *Not being used at this time "=0"*

P607(8) *Not being used at this time "=0"*
(7) Status report by interruption (0=No, 1=Yes) ..... *Section 4.3.4*
(6) CNC aborts DNC communications (program debugging) (1=No, 0=Yes) ..... *Section 4.3.4*
(5) DNC protocol active on power-up (0=No, 1=Yes) ..... *Section 4.3.4*
(4) Communications setting. FAGOR cassette (0) FAGOR Floppy Disk Unit (1) ... *Section 4.3.4*
(3) DNC (0=No, 1=Yes) ..... *Section 4.3.4*
(2) *Not being used at this time "=0"*
(1) *Not being used at this time "=0"*

P608 *Not being used at this time "=0"*

P609(8) Axis orientation for graphic display (0=Mill, 1=Boring Mill) ..... *Section 4.3.3*
(7) Pin 17 of connector I/O 1 as "rapid jog" (0=No, 1=Yes) ..... *Section 4.3.1*
(6) Maximum incremental JOG move (0=10mm or 1", 1=1mm or 0.1") ..... *Section 4.3.3*
(5) M functions set at M table are output in BCD or binary (1=No, 0=Yes) ..... *Section 4.3.1*
(4) All spindle speed changes generate an S strobe output (0=No, 1=Yes) ..... *Section 6.2*
(3) Pin 23 of connector I/O 2 as "RESET" (0=No, 1=Yes) ..... *Section 4.3.1*
(2) *Not being used at this time "=0"*
(1) Machine with axis travel over 8 meters (0=No, 1=Yes) ..... *Section 5.9*

P610(8) X axis binary encoder equivalence ..... *Section 5.1*
(7) Y axis binary encoder equivalence ..... *Section 5.1*
(6) Z axis binary encoder equivalence ..... *Section 5.1*
(5) *Not being used at this time "=0"*
(4) S analog output: unipolar (1) or bipolar (0) ..... *Section 6.2*
(3) *Not being used at this time "=0"*
(2) *Not being used at this time "=0"*
(1) Feed-Hold in G47 (0=No, 1=Yes) ..... *Section 4.3.3*

P611(8) In G00 and F00 proportional gain break-point at 256 µm (0=No, 1=Yes) ..... *Section 5.5*
(7) *Not being used at this time "=0"*
(6) Pin 24 of I/O 2 as "P99996 in execution" output (0=No, 1=Yes) ..... *Section 4.3.1*
(5) Feedrate units in G94 (0=1mm/min or 0.1"/min, 1=0.1mm/min or 0.01/min") .. *Section 4.3.3*
(4) *Not being used at this time "=0"*
(3) *Not being used at this time "=0"*
(2) Home search required after power-up (0=No, 1=Yes) ..... *Section 5.6*
(1) Pin 24 of I/O 2 as "Program block in execution" output (0=No, 1=Yes) ..... *Section 4.3.1*

P612(8) *Not being used at this time "=0"*
(7) *Not being used at this time "=0"*
(6) Multplying factor for X axis electronic handwheel signals (0=x4, 1=x2) ..... *Section 4.3.2*
(5) Feedback (counting) resolution of the X axis electronic handwheel ..... *Section 4.3.2*
(4) Feedback (counting) resolution of the X axis electronic handwheel ..... *Section 4.3.2*
(3) Feedback units of the X axis electronic handwheel (0=mm, 1=inch) ..... *Section 4.3.2*
(2) Counting direction of the X axis electronic handwheel ..... *Section 4.3.2*
(1) There is an electronic handwheel associated with the X axis (0=No, 1=Yes) .... *Section 4.3.2*

P613(8)
(7) Acc/dec in all linear interpolations (0=No, 1=Yes) ..... *Section 5.7*
(6) *Not being used at this time "=0"*
(5) G05 or G07 active on power-up (0=G07, 1=G05) ..... *Section 4.3.3*
(4) Pin 24 of connector I/O 2 as "rapid traverse" output (0=No, 1=Yes) ..... *Section 4.3.1*
(3) *Not being used at this time "=0"*
(2) Pin 25 of connector I/O 2 as "vertical move" output ..... *Section 4.3.1*
(1) Electronic handwheel model: FAGOR 100P (0=No, 1=Yes) ..... *Section 4.3.2*

P614 *Not being used at this time "=0"*

P615(8,7) *Not being used at this time "=0"*
(6) Feedrate in 0.1inch/min (0) or inches/min. (1) ..... *Section 5.4*
(5,4,3,2,1) *Not being used at this time "=0"*

P616 *Not being used at this time "=0"*

P617(8) M function output in BCD (0) or Binary (1) ..... *Section 4.3.1*
(7) *Not being used at this time "=0"*
(6) 0.0001 millimeters (0.00001 inch) resolution (0=No, 1=Yes) ..... *Section 5.9*
(5) X as DRO axis (0=No, 1=Yes) ..... *Section 4.3*
(4) Z as DRO axis (0=No, 1=Yes) ..... *Section 4.3*
(3,2,1) *Not being used at this time "=0"*

P618(8) *Not being used at this time "=0"*
(7) *Not being used at this time "=0"*
(6) X axis display (1=No, 0=Yes) ..... *Section 4.3*
(5) Y axis display (1=No, 0=Yes) ..... *Section 4.3*
(4) Z axis display (1=No, 0=Yes) ..... *Section 4.3*
(3) *Not being used at this time "=0"*
(2) *Not being used at this time "=0"*
(1) CYCLE START key inhibited (0=No, 1=Yes) ..... *Section 4.3.3*

P619 *Not being used at this time "=0"*

P620(8,7,6,5,4,3) *Not being used at this time "=0"*
(2) Acc/dec. in G05 (round corner) (1=No, 0=Yes)..... *Section 5.7*
(1) *PLCI marks M1801..M1899 to send messages to CNC* (0=No, 1=Yes) ... *PLCI manual*

P621(8,7,6,5,4,3,2) *Not being used at this time "=0"*
(1) The CNC has a PLCI (0=No, 1=Yes) ..... PLCI manual

P622(8,7,6,5,4) *Not being used at this time "=0"*
(3) Z axis sine-wave feedback resolution ..... *Section 5.1*
(2) Y axis sine-wave feedback resolution ..... *Section 5.1*
(1) X axis sine-wave feedback resolution ..... *Section 5.1*

P623(8) X axis home searching direction (0=Positive, 1=Negative)..... *Section 5.6*
(7) Y axis home searching direction (0=Positive, 1=Negative)..... *Section 5.6*
(6) Z axis home searching direction (0=Positive, 1=Negative) ..... *Section 5.6*
(5,4,3,2,1) *Not being used at this time "=0"*

P624(8) Bell shaped acc/dec. (0=No, 1=Yes) ..... *Section 5.7*
(7) *Not being used at this time "=0"*
(6) *Not being used at this time "=0"*
(5) *Not being used at this time "=0"*
(4) *Not being used at this time "=0"*
(3) Sign of Z axis leadscrew backlash (0=Positive, 1=Negative) ..... *Section 5.8*
(2) Sign of Y axis leadscrew backlash (0=Positive, 1=Negative)..... *Section 5.8*
(1) Sign of X axis leadscrew backlash (0=Positive, 1=Negative)..... *Section 5.8*

P625 *Not being used at this time "=0"*

P626(8) Electonic handwheel associated with the Y axis (0=No, 1=Yes). ..... Section 4.3.2
(7) *Not being used at this time "=0"*
(6) Multplying factor for Y axis electronic handwheel (0=x4, 1=x2) ..... *Section 4.3.2*
(5) Feedback (counting) resolution of the Y axis electronic handwheel ..... *Section 4.3.2*
(4) Feedback (counting) resolution of the Y axis electronic handwheel ..... *Section 4.3.2*
(3) Feedback units of the Y axis electronic handwheel (0=mm, 1=inch)..... *Section 4.3.2*
(2) Counting direction of the Y axis electronic handwheel ..... *Section 4.3.2*
(1) The CNC displays the tool base (0) or tool tip (1) position ..... *Section 4.3*

P627(8) Electonic handwheel associated with the Z axis (0=No, 1=Yes). ..... Section 4.3.2
(7) *Not being used at this time "=0"*
(6) Multplying factor for Z axis electronic handwheel (0=x4, 1=x2) ..... *Section 4.3.2*
(5) Feedback (counting) resolution of the Z axis electronic handwheel..... *Section 4.3.2*
(4) Feedback (counting) resolution of the Z axis electronic handwheel..... *Section 4.3.2*
(3) Feedback units of the Z axis electronic handwheel (0=mm, 1=inch) ..... *Section 4.3.2*
(2) Counting direction of the Z axis electronic handwheel ..... *Section 4.3.2*
(1) *Not being used at this time "=0"*

P628(8) The machine uses mechanical handwheels (0=No, 1=Yes) ..... Section 4.3.2
(7) Z axis interpolation (1=No, 0=Yes) ..... Section 4.3.3
(6) O1 may be turned on/off only when spindle is off (0=No, 1=Yes) ..... Section 4.3
(5) Electronic handwheels only operate at handwheel positions (1=No, 0=Yes) .... Section 4.3.2
(4) The rapid jog key applies a feedrate override over 100% (0=No, 1=Yes) ..... Section 4.3.3
(3) *Not being used at this time "=0"*
(2) *Not being used at this time "=0"*
(1) Display of the following error (0=No, 1=Yes) ..... Section 4.3

P629 thru P635 *Not being used at this time "=0"*

P700 *Not being used at this time "=0"*
P701 Number of tools (0. 98) ..... *Section 4.3*
P702 thru P705 *Not being used at this time "=0"*
P706 Value of the residual S analog voltage (1=2.5mV) ..... *Section 6.1*
P707 Oscillation period during range change ..... *Section 6.1*
P708 Feedrate override When analog voltage reaches 10V. ..... *Section 5.4*
P709 thu P713 *Not being used at this time "=0"*
P714 Error if actual feedrate not within 50% & 200% of programmed value . *Section 5.4*
P715 Dwell between blocks in G07 (square corner) (1=10ms) ..... *Section 4.3.8*
P716 thru P720 *Not being used at this time "=0"*
P721 ACCELERATION/DECELERATION for X (1=20ms) ..... *Section 5.7*
P722 ACCELERATION/DECELERATION for Y (1=20ms) ..... *Section 5.7*
P723 ACCELERATION/DECELERATION for Z (1=20ms) ..... *Section 5.7*
P724, P725 *Not being used at this time "=0"*
P726 Recovery of programmed position on axes "non-continuously control" *Section 5.5*
P727, P728 *Not being used at this time "=0"*
P729 Maximum feedrate for circular interpolations ..... *Section 5.4*
P730, P731 *Not being used at this time "=0"*
P732 FEED-FORWARD gain for X ..... *Section 5.7*
P733 FEED-FORWARD gain for Y ..... *Section 5.7*
P734 FEED-FORWARD gain for Z ..... *Section 5.7*
P735 thru P740 *Not being used at this time "=0"*
P741 Execution frequency for main PLCI program cycle ..... PLCI manual
P742 *Not being used at this time "=0"*
P743 Subroutine to be execued before the T function ..... Section 4.3
P744 Bell shaped acc/dec. ramp duration (1=10ms) ..... *Section 5.7*
P745 Subroutine to be execued after the T function ..... Section 4.3
P750 *Not being used at this time "=0"*

P800 thru P803 *Not being used at this time "=0"*
P804 Jogging feedrate prior to mandatory home search ..... *Section 5.6*
P805, P809 *Not being used at this time "=0"*
P810 2nd home searching feedrate for X ..... *Section 5.6*
P811 2nd home searching feedrate for Y ..... *Section 5.6*
P812 2nd home searching feedrate for Z ..... *Section 5.6*
P813, P814 *Not being used at this time "=0"*
P815 Delay before opening the closed loop (1=10ms) ..... *Section 4.3.2*

P900 thru P907 *Not being used at this time "=0"*
P908 Collision zone between Y, Z ..... *Section 5.9*
P909 Collision zone between Y, Z ..... *Section 5.9*
P910 thru P918 *Not being used at this time "=0"*

APPENDIX G

MACHINE PARAMETER SETTING CHART

	VALUE		VALUE		VALUE		VALUE		VALUE
P0		P1		P2		P3		P4	
P5		P6		P7		P8		P9	
P10		P11		P12		P13		P99	

	VALUE		VALUE		VALUE		VALUE		VALUE
P100		P101		P102		P103		P104	
P105		P106		P107		P108		P109	
P110		P111		P112		P113		P114	
P115		P116		P117		P118		P119	

P200		P201		P202		P203		P204	
P205		P206		P207		P208		P209	
P210		P211		P212		P213		P214	
P215		P216		P217		P218		P219	

P300		P301		P302		P303		P304	
P305		P306		P307		P308		P309	
P310		P311		P312		P313		P314	
P315		P316		P317		P318		P319	

	VALUE		VALUE		VALUE		VALUE		VALUE
P600		P601		P602		P603		P604	
P605		P606		P607		P608		P609	
P610		P611		P512		P613		P614	
P615		P616		P617		P618		P619	
P620		P621		P622		P623		P624	
P625		P626		P627		P628		P629	
P630		P631		P632		P633		P634	
P635									

	VALUE		VALUE		VALUE		VALUE		VALUE
P700		P701		P702		P703		P704	
P705		P706		P707		P708		P709	
P710		P711		P712		P713		P714	
P715		P716		P717		P718		P719	
P720		P721		P722		P723		P724	
P725		P726		P727		P728		P729	
P730		P731		P732		P733		P734	
P735		P736		P737		P738		P739	
P740		P741		P742		P743		P744	
P745		P746		P747		P748		P749	
P750		P751		P752		P753			

	VALUE		VALUE		VALUE		VALUE		VALUE
P800		P801		P802		P803		P804	
P805		P806		P807		P808		P809	
P810		P811		P812		P813		P814	
P815		P816		P817		P818		P819	
P820		P821		P822		P823			

	VALUE		VALUE		VALUE		VALUE		VALUE
P900		P901		P902		P903		P904	
P905		P906		P907		P908		P909	
P910		P911		P912		P913		P914	
P915		P916		P917		P918		P919	
P920		P921		P922		P923		P924	
P925		P926		P927		P928		P929	
P930		P931		P932		P933		P934	
P935									

APPENDIX H

DECODED "M" FUNCTION SETTING CHART

Auxiliary Function	VALUE
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	
M	

APPENDIX I

LEADSCREW ERROR COMPENSATION SETTING CHART

X AXIS			
Position		**ERROR**	
P0		P1	
P2		P3	
P4		P5	
P6		P7	
P8		P9	
P10		P11	
P12		P13	
P14		P15	
P16		P17	
P18		P19	
P20		P21	
P22		P23	
P24		P25	
P26		P27	
P28		P29	
P30		P31	
P32		P33	
P34		P35	
P36		P37	
P38		P39	
P40		P41	
P42		P43	
P44		P45	
P46		P47	
P48		P49	
P50		P51	
P52		P53	
P54		P55	
P56		P57	
P58		P59	

Y AXIS			
Position		**ERROR**	
P60		P61	
P62		P63	
P64		P65	
P66		P67	
P68		P69	
P70		P71	
P72		P73	
P74		P75	
P76		P77	
P78		P79	
P80		P81	
P82		P38	
P84		P85	
P86		P87	
P88		P89	
P90		P91	
P92		P93	
P94		P95	
P96		P97	
P98		P99	
P100		P101	
P102		P103	
P104		P105	
P106		P107	
P108		P109	
P110		P111	
P112		P113	
P114		P115	
P116		P117	
P118		P119	

Z AXIS			
Position		**ERROR**	
P120		P121	
P122		P123	
P124		P125	
P126		P127	
P128		P129	
P130		P131	
P132		P133	
P134		P135	
P136		P137	
P138		P139	
P140		P141	
P142		P143	
P144		P145	
P146		P147	
P148		P149	
P150		P151	
P152		P153	
P154		P155	
P156		P157	
P158		P159	
P160		P161	
P162		P163	
P164		P165	
P166		P167	
P168		P169	
P170		P171	
P172		P173	
P174		P175	
P176		P177	
P178		P179	

APPENDIX J

MAINTENANCE

Cleaning:

The accumulated dirt inside the unit may act as a screen preventing the proper dissipation of the heat generated by the internal circuitry which could result in a harmful overheating of the CNC and, consequently, possible malfunctions.

On the other hand, accumulated dirt can sometimes act as an electrical conductor and shortcircuit the internal circuitry, especially under high humidity conditions.

To clean the operator panel and the monitor, a smooth cloth should be used which has been dipped into de-ionized water and /or non abrasive dish-washer soap (liquid, never powder) or 75º alcohol.

Do not use highly compressed air to clean the unit because it could generate electrostatic discharges.

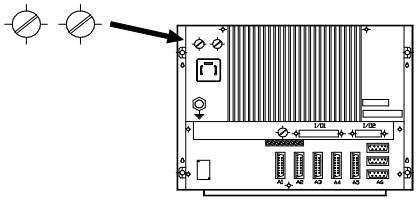
The plastics used on the front panel of the CNC are resistant to :

1.- Grease and mineral oils
2.- Bases and bleach
3.- Dissolved detergents
4.- Alcohol

Avoid the action of solvents such as Chlorine hydrocarbons , Benzole , Esters and Ether which can damage the plastics used to make the unit's front panel.

Preventive Inspection:

If the CNC does not turn on , verify that the fuses are in good condition and that they are the right ones.



It carries two fast fuses of 3.15 Amp./ 250V. to protect the mains AC input.

To check the fuses, first disconnect the power to the CNC.

Do not manipulate inside this unit.

Only personnel authorized by Fagor Automation may manipulate inside this module.

Do not manipulate the connectors with the unit connected to main AC power.

Before manipulating these connectors, make sure that the unit is not connected to main AC power.

Note :

Fagor Automation shall not be held responsible for any material or physical damage derived from the violation of these basic safety requirements.

List of materials, parts that could be replaced

Part Description		*Code*	*Manufacturer*	*Reference*
Mains cable 3x0.75		11313000	Fagor Automation	
3.15A/250V Fuse		12130015	Schurter Wickmann	FST-034-1521 Ref. 19115
English manual	OEM USER	83750033 83750022	Fagor Automation	

ERROR CODES

001 This error occurs in the following cases:

* When the first character of the block to be executed is not an "N".

* When while BACKGROUND editing, the program in execution calls a subroutine located in the program being edited or in a later program.

  The order in which the part-programs are stored in memory are shown in the part-program directory. If during the execution of a program, a new one is edited, this new one will be placed at the end of the list.

002 Too many digits when defining a function in general.

003 This error occurs in the following cases:

* When a negative value has been assigned to a function which does not accept the "-" sign.

* When an incorrect value has been assigned to an automatic operation:

-	Positioning in line:	If	L=0, Xn=X1, Yn=Y1, I=0		
		If	L=0, Xn=X1, Yn=Y1, N=0		
		If	I=0, N=0		
		If	I>0, L/I fraction		
-	Positioning in arc:	If	N=0		
		If	R=0, Xc=X1, Yc=Y1		
-	Positioning in rectangle or grid pattern:	If	LX=0, IX=0	or	LY=0, IY=0
		If	LX=0, NX=0	or	LY=0, NY=0
		If	LX>0, IX=0, NX<2	or	LY>0, IY=0, NY<2
		If	LX>0, IX>0, LX/IX fraction		
		If	LY>0, IY>0, LY/IY fraction		
-	Rectangular pocket	If	L=0 or H=0		
		If	r>(L/2) or r>(H/2)		
-	Circular pocket	If	Tool radius > R		
-	Corner roughing	If	L=0 or H=0		
		If	r>L or r>H		
-	Surface milling	If	L=0 or H=0		

004 Not being used at this time.

005 Parametric block programmed wrong.

006 There are more than 10 parameters affected in a block.

007 Division by zero.

008 Square root of a negative number.

009 Parameter value too large.

010 M41, M42, M43 or M44 has been programmed.

011 More than 7 "M" functions in a block.

012 This error occurs in the following cases:

- Function G50 is programmed wrong

- Tool dimension values too large.

- Zero offset values (G53/G59) too large.

013 Not being used at this time.

014 A block has been programmed which is incorrect either by itself or in relation with the program history up to that instant.

015 Functions G20, G21, G22, G23, G24, G25, G26, G27, G28, G29, G30, G31, G32, G50, G52, G53, G54, G55, G56, G57, G58, G59, G72, G73, G74, G92 and G93 must be programmed alone in a block.

016 The called subroutine or block does not exist or the block searched by means of special function F17 does not exist.

017 Negative or too large thread pitch value.

018 Error in blocks where the points are defined by means of angle-angle or angle-coordinate.

019 This error is issued in the following cases:

- After defining G20, G21, G22 or G23, the number of the subroutine it refers to is missing.
- The "N" character has not been programmed after function G25, G26, G27, G28 or G29.
- Too many nesting levels.

020 The axes of the circular interpolation are not programmed correctly.

021 There is no block at the address defined by the parameter assigned to F18, F19, F20, F21, F22.

022 An axis is repeated when programming G74.

023 K has not been programmed after G04.

025 Error in a definition block or subroutine call, or when defining either conditional or unconditional jumps.

026 This error is issued in the following cases:

- Memory overflow.
- Not enough free tape or CNC memory to store the part-program.

027 I/J/K has not been defined for a circular interpolation or thread.

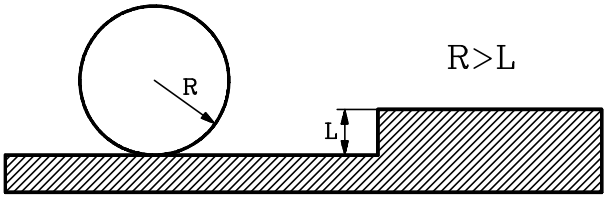
028 An attempt has been made to select a tool offset at the tool table or a non-existent external tool (the number of tools is set by machine parameter).

029 Too large a value assigned to a function.

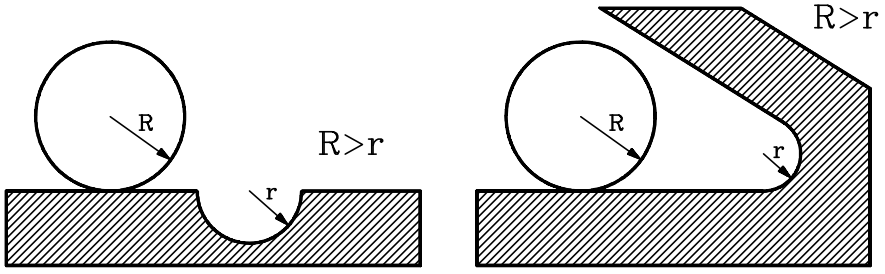
This error is often issued when programming an F value in mm/min (inch/min) and, then, switching to work in mm/rev (inch/rev) without changing the F value.

030 The programmed G function does not exist.

031 Tool radius value too large.



032 Tool radius value too large.



033 A movement of over 8388 mm or 330.26 inches has been programmed.

Example: Being the X axis position X-5000, if we want to move it to point X5000, the CNC will issue error 33 when programming the block N10 X5000 since the programmed move will be:
5000 - (-5000) = 10000 mm.

In order to make this move without issuing this error, it must be carried out in two stages as indicated below:

```
N10 X0          ; 5000 mm move
N10 X5000       ; 5000 mm move
```

034 S or F value too large.

035 Not enough information for corner rounding, chamfering or compensation.

036 Repeated subroutine.

037 Function M19 programmed incorrectly.

038 Function G72 or G73 programmed incorrectly.

It must be borne in mind that if G72 is applied only to one axis, this axis must be positioned at part zero (0 value) at the time the scaling factor is applied.

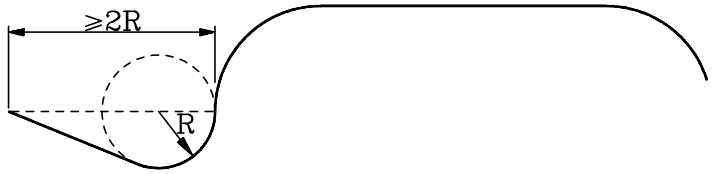
039 This error occurs in the following cases:

- More than 15 nesting levels when calling subroutines.
- A block has been programmed which contains a jump to itself. Example: N120 G25 N120.

040 The programmed arc does not go through the defined end point (tolerance 0.01mm) or there is no arc that goes through the points defined by G08 or G09.

041 This error is issued when programming a tangential entry as in the following cases:

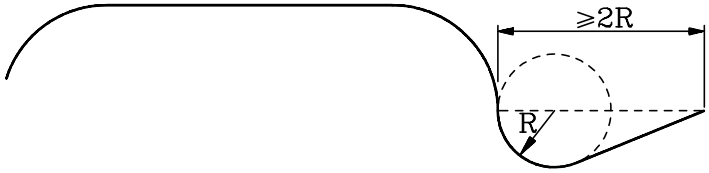
- There is no room to perform the tangential entry. A clearance of twice the rounding radius or greater is required.



- If the tangential entry is to be applied to an arc (G02, G03), The tangential entry must be defined in a linear block.

042 This error is issued when programming a tangential exit as in the following cases:

- There is no room to perform the tangential exit. A clearance of twice the rounding radius or greater is required.



- If the tangential exit is to be applied to an arc (G02, G03), The tangential exit must be defined in a linear block.

043 Polar origin coordinates (G93) defined incorrectly.

044 Not being used at this time.

045 Function G36, G37, G38 or G39 programmed incorrectly.

046 Polar coordinates defined incorrectly.

047 A zero movement has been programmed during radius compensation or corner rounding.

048 Not being used at this time.

049 Chamfer programmed incorrectly in a rectangular pocket or corner roughing operation in such way that:

* The tool cannot machine it because the chamfer is too small.
* A chamfer that big cannot be machined with those L, H, E parameter values

050 Functions M06, M22, M23, M24, M25 must be programmed alone in a block.

051 * A tool change cannot be performed without being in the change position.

052 * The requested tool is not in the magazine.

053 Not being used at this time.

054 There is no tape in the cassette reader or the reader head cover is open.

055 Parity error when reading or recording a cassette.

056 Not being used at this time.

057 Write-protected tape.

058 Sluggish tape transport.

059 Communication error between the CNC and the cassette reader.

060 Internal CNC hardware error. Consult with the Technical Service Department.

061 Battery error.

The memory contents will be kept for 10 more days (with the CNC off) from the moment this error occurs. The whole battery module located on the back must be replaced. Consult with the Technical Service Department.

Atention:

Due to danger of explosion or combustion, do not try to recharge the battery, do not expose it to temperatures higher than 100°C (232°F) and do not short the battery leads.

064 * External emergency input (pin 14 of connector I/O1) is activated.

065 Not being used at this time.

066 * X axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

067 * Y axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

068 * Z axis travel limit overrun.

It is generated either because the machine is beyond limit or because a block has been programmed which would force the machine to go beyond limits.

069 Not being used at this time.

070 ** X axis following error.

071 ** Y axis following error.

072 ** Z axis following error.

073 Not being used at this time.

074 ** Spindle speed value too large.

075 ** Feedback error at connector A1.

076 ** Feedback error at connector A2.

077 ** Feedback error at connector A3.

078 ** Feedback error at connector A4.

079 ** Feedback error at connector A5.

080 This error occurs when using a tool smaller than the machining pass "G" in a rectangular/circular pocket or in a corner roughing operation.

081 This error occurs when the tool radius is greater than "(L/2)-E" or "H/2)-E".

082 ** Parity error in general parameters.

083 This error occurs when programming "r>0" or "C>0" in a rectangular pocket or corner roughing operation.

084 This error occurs when programming a tool radius greater than "R-E" in a circular pocket.

085 This error occurs when using a 0-radius tool (tool offset) having programmed "G=0" (machining pass) in a rectangular/circular pocket or in a corner roughing operation.

086 This error occurs when assigning an incorrect value to an automatic operation or to a machining operation:

- Rectangular pocket .....If P=0 or I=0
- Circular pocket .....If P=0 or I=0
- Corner roughing.....If P=0 or I=0
- Surface milling .....If P=0 or I=0
- Center punching: .....If P=0, Φ =0
- Drilling: .....If P=0 or I=0
- Tapping: .....If P=0
- Boring, reaming: .....If P=0

087 ** Internal CNC hardware error. Consult with the Technical Service Department.

088 ** Internal CNC hardware error. Consult with the Technical Service Department.

089 * All the axes have not been homed.

This error comes up when it is mandatory to search home on all axes after power-up. This requirement is set by machine parameter.

090 ** Internal CNC hardware error. Consult with the Technical Service Department.

091 ** Internal CNC hardware error. Consult with the Technical Service Department.

092 ** Internal CNC hardware error. Consult with the Technical Service Department.

093 ** Internal CNC hardware error. Consult with the Technical Service Department.

094 Parity error in tool table or zero offset table G53-G59.

095 This error occurs when the tool radius is larger than the rounding radius "r" in a rectangular pocket or corner roughing operation.

096 ** Parity error in Z axis parameters.

097 ** Parity error in Y axis parameters.

098 ** Parity error in X axis parameters.

099 ** Parity error in M table.

100 ** Internal CNC hardware error. Consult with the Technical Service Department.

101 ** Internal CNC hardware error. Consult with the Technical Service Department.

105 This error comes up in the following cases:

> A comment has more than 43 characters.

> A program has been defined with more than 5 characters.

> A block number has more than 4 characters.

> Strange characters in memory.

106 ** Inside temperature limit exceeded.

107 Not being used at this time.

108 ** Error in Z axis leadscrew error compensation parameters.

109 ** Error in Y axis leadscrew error compensation parameters.

110 ** Error in X axis leadscrew error compensation parameters.

111 Not being used at this time.

112 Not being used at this time.

113 Not being used at this time.

114 Not being used at this time.

115 * Watch-dog error in the periodic module.

This error occurs when the periodic module takes longer than 5 milliseconds.

116 * Watch-dog error in the main module.

This error occurs when the main module takes longer than half the time indicated in machine parameter "P729".

117 * The internal CNC information requested by activating marks M1901 thru M1949 is not available.

118 * An attempt has been made to modify an unavailable internal CNC variable by means of marks M1950 thru M1964.

119 Error when writing machine parameters, the decoded M function table and the leadscrew error compensation tables into the EEPROM memory.

This error may occur when after locking the machine parameters, the decoded M function table and the leadscrew error compensation tables, one tries to save this information into the EEPROM memory.

120 Checksum error when recovering (restoring) the machine parameters, the decoded M function table and leadscrew error compensation tables from the EEPROM memory.

Atention:

The ERRORS indicated with "*" behave as follows:

They stop the axis feed and the spindle rotation by cancelling the Enable signals and the analog outputs of the CNC.

hey interrupt the execution of the part-program of the CNC if it was being executed.

The ERRORS indicated with "**" besides behaving as those with an "*", they activate the INTERNAL EMERGENCY OUTPUT.